IO1 – Methodology for defining 3D printing exercises suitable for transversal education

3D PRINTING TECHNICAL GUIDE

- O1A1-
Transfer of knowledge about basics of 3D printing concepts to the VET teachers.

E3D+VET

ERASMUS+
3D PRINTING
VET CENTRES

ERASMUS3D+
For the immersion in 3D printing of VET centres.

Project Agreement Number
2017-1-DE02-KA202-004159

PARTICIPANT ORGANIZATIONS

KIT – Karlsruhe Institute of Technology

Technical Research Centre of Furniture and Wood of the Region of Murcia — ceTEM

STYRIAN TECHNOLOGY PARK
Regional Centre for Technology Development
Business Support Centre and Incubator

SEMINAR
BERUFLICHE SCHULEN
KARLSRUHE

CEIPES

pristalica _software studio

Disclaimer:

Output Identification	O1
Output Title	IO1 – Methodology for defining 3D printing exercises suitable for transversal education
Output Description	**O1 – A1. Transfer of knowledge about basic of 3D printing concepts to the VET teachers.**
Version	**v.1.1 (Long Version)**

INDEX

FIGURE INDEX

SCHEMES INDEX

TABLES INDEX

1. INTRODUCTION TO ADDITIVE MANUFACTURING

1.1. What is Additive Manufacturing?

Additive Manufacturing or 3D printing (commonly called) is a process that creates a physical object from a digital design. There are different 3D printing technologies and materials you can print with, but all are based on the same principle: a digital model is turned into a solid three-dimensional physical object by adding material layer by layer. [1]

It is important to point out from the beginning that Additive Manufacturing does not constitute a single technology but a set of manufacturing processes, very different from each other, that share three common characteristics:

1. They are manufacturing processes by addition of material to construct a solid three-dimensional object.
2. The object is constructed by superimposing successive layers of material.
3. The object is made from a digital 3D model.

They are called ADITIVE Manufacturing processes to differentiate them from conventional processes. Together with these, they are part of the set of processes available to the Industry.
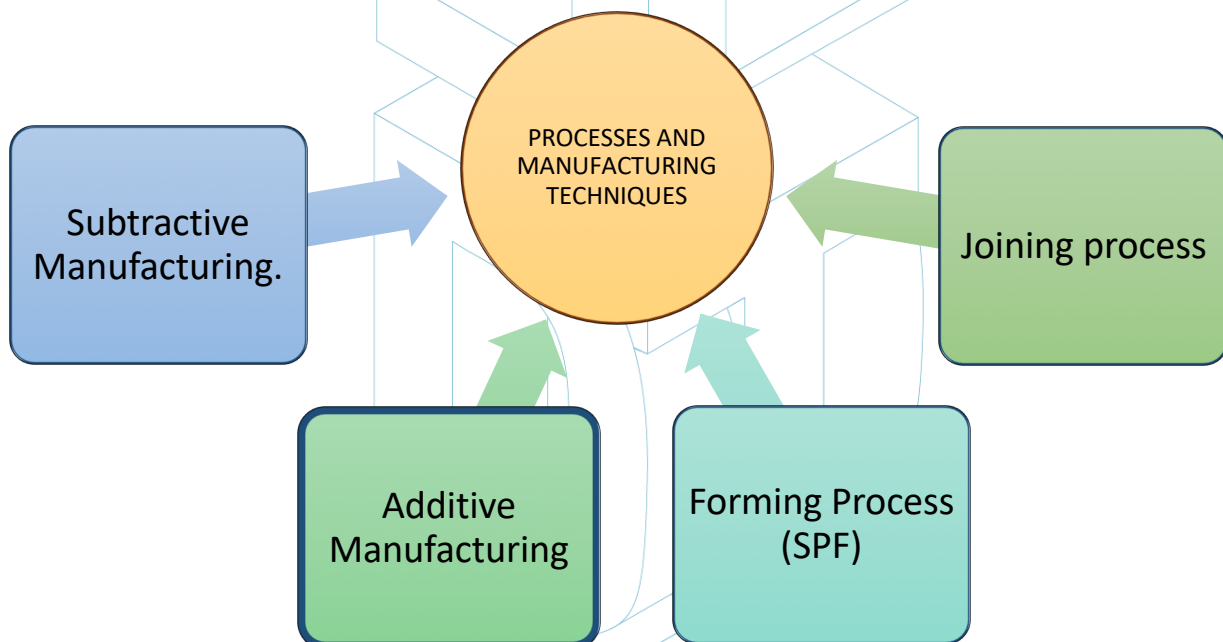
Scheme 1: Processes and manufacturing techniques. [2]

Some of the most used additive manufacturing technologies that best suits to the educational area will be described in the following point of this guide. These technologies are: Fused Deposition Modeling (FDM), Stereolithographic (SLA) and Selective Laser Sintering (SLS).

Figure 1: FDM Printer. [3]

Figure 2: SLA Printer. [4]
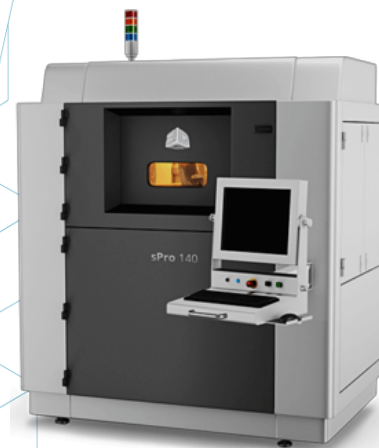
Figure 3: SLS Printer. [5]

1.2.   How does 3D printing work?

It all starts with making or obtaining a virtual design of the object you want to create. This virtual design can be made in a CAD (Computer Aided Design) file using a 3D modeling program (for the creation of a totally new object) or with the use of a 3D scanner (to copy an existing object). A 3D scanner makes a 3D digital copy of an object. There are also lots of online file repositories where you can download existing 3D files that will help get you started.

The 3D printing process turns an object into many, tiny little slices, then builds it from the bottom up, slice by slice. The layers then build up to form a solid object. [6]
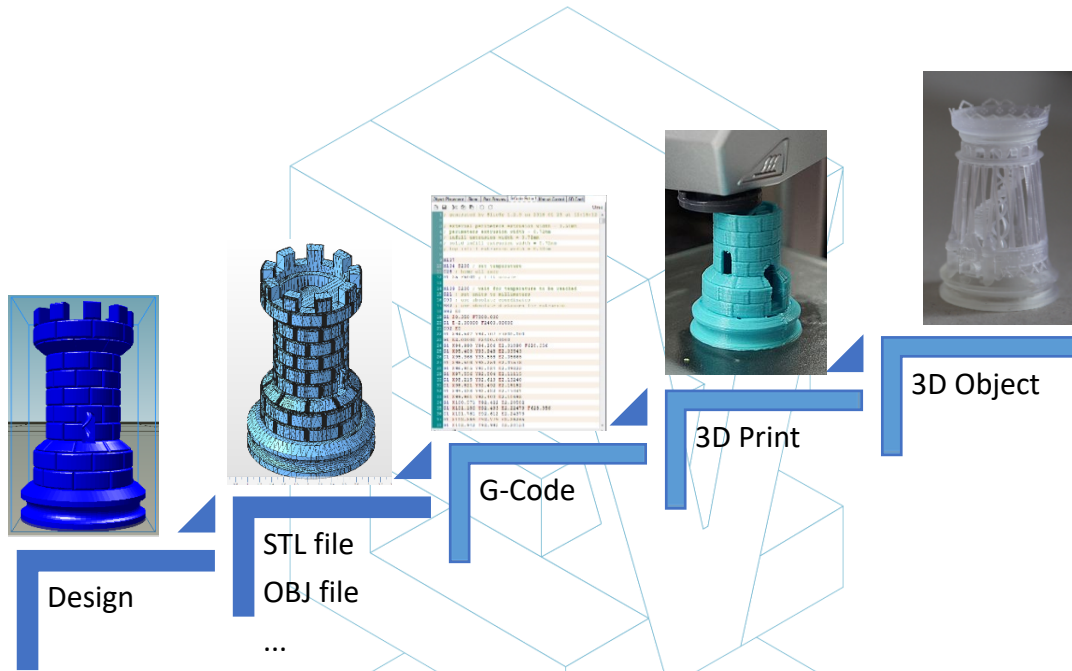
Figure 4: 3D printing process [6]

The full process is explained in the point 3 of this guide.

Some advantages of Additive Manufacturing compared to conventional processes:

- Fewer steps between the CAD model and the production of the part.
- Generally, few human resource requirements due to a high level of automation.
- A large number of geometrical forms can be manufactured, enabling for instance the production of parts which are topologically optimised, with internal channels, etc.
- High-speed manufacturing for small, complex parts.
- Generally, less material wastage.
- Possibility to reconstruct damaged sections of existing objects, depending on the part material
- No special tooling required.

1.3. What is Rapid Prototyping?

Rapid prototyping is an automated process that quickly builds physical prototypes from 3D CAD files composed of surface quality or solid models. Any manufacturing process can be classified as either subtractive, formative or additive. Every manufacturing process either falls completely into one of these categories or is a hybrid process falling into more than one. In the manufacturing arena, productivity is achieved by guiding a product from concept to market quickly and inexpensively. Rapid prototyping technology aids this process. [8]

It is important not to confuse rapid prototyping with 3D printing or with additive manufacturing, because the concepts are used interchangeably and wrongly many times. We can say that additive manufacturing is one of the technologies with which we can produce a rapid prototyping product.

It is convenient to underline that every technology and every process has a starting point in common: computer aided design (CAD).

Here some of the most common techniques for Rapid Prototyping:

- 3D Scanner / Reverse Engineering.
- Additive Manufacturing.
- CNC Machining.
- Vacuum Fast Casting.
- Prototype Moulds.
- Sand Casting.
- Investment Casting.

The purpose of rapid prototyping is to test the various design features, ideas, concepts, functionality, output and performance.

2. TECHNOLOGIES

In order to identify the most suitable 3D printing technologies for educational purposes, the following table was developed. It presents a comparison between a total of 7 technologies, which are the most used currently, according to the latest trends. The analysis was based on 12 parameters, which are considered the best ones in order to have a full overview on what 3D printing offers. Special focus was given to strengths and weaknesses for each technology as they are important indicators in the identification.

Figure 5: Prototypes of a ski goggles' frame printed with FDM, SLA and SLS technology (from left to right). [9]

Through the study carried out in the report IO1 / A3: Identification of 3D printing most suitable technologies for education, the consortium identified the three printing technologies described below as the most recommended for use in the educational field. Here, in addition to the Process, Materials, Application Areas, the Strengths and Weaknesses are also described for each one of them.

TECHNOLOGIES	Process	Materials used	Complexity	Speed	Max Part Size (cm)	Accuracy	Surface Finish	Strengths	Weaknesses	Pricing	Application Area	Application Examples
Fused Deposition Modeling (FDM)	Layers of melted plastic	ABS Filaments, Polycarbonate, Resin, Nylon	•••	Fair	30x30x50	Fair	Fair	Durable; ideal for conceptual models	Low resolution	€€	Aerospace, automotive, industrial, medical	Wind turbines, aircraft components
Selective Laser Sintering (SLS)	Plastic powder melted by laser	Paper, plastic, metal, glass, ceramic, composites	•••	Fast	34x34x60	Good	Fair	Resistant, durable, flexible	Needs post-processing	€€	Automotive, consumer products, aerospace	Small production batches and prototypes
Stereolithography (SLA)	Polymerization scanned by UV laser	Liquid photopolymer, composites	•••	Fast	30x30x50	Very good	Very good	High res; complex geometries	Only photopolymer materials	€€€	Aerospace, automotive, consumer goods	Medical models of anatomic human parts
Photopolymer Jetting (POLYJET)	Inkjet method with liquid photopolymers	Metals, plastic, wax	•••	Fast	39x31x19	Very good	Good	More materials at the same time	Only photopolymer materials; not durable	€€€	Medical devices, multimaterial prototypes	Medical stethoscopes
Selective Laser Melting (SLM)	Metal powder melted by laser	Metals: copper, aluminium, tungsten etc.	••	Fair	28x28x36	Fair	Fair	Manufactures high density parts	Price; needs post-processing	€€	Dental products, mechanical components	Lightweight components for aircraft
Electron Beam Melting (EBM)	Melted powder selected by electron beam	Metals: cobalt, chrome, nickel	•••	Fast	20x20x20	Fair	Poor	Less thermal stress	Limited set of metals	€€€	Dental, medical implants, automotive	Bone tissue medical models
Electron Binder Jetting (BJ)	Powder distributed by jetting machine	Ceramic, metals, plastic, sand, composite	•	Fast	40x20x10	Fair	Fair	No support structure; multicolour prints	Fragile with limited mechanical properties	€	Architecture, mechanical structures	Pots and general home furniture
Continuous Fibre Fabrication (CFF)	Double nozzle laying/melting method	Plastic, carbon composites, nylon	••••	Fair	32x43x16	Fair	Fair	Robust parts, no post-process needed	Limited fibre placement	€€€	Aerospace	Lightweight components
Material Jetting (MJ)	Inkjet method with wax materials	Wax	••	Slow	30x18x20	Very good	Good	High resolution	Limited wax-like materials; requires support structure	€€	Prototypes for form, fit testing; Casting patterns	Lost Wax Casting in Jewellery and Medical fields

Figure 6: 3DP Technologies Comparison. [10]

LEGEND:
• = Simple;   € = Cheap;
•• = Fair;   €€ = Fair;
••• = Complex;   €€€ = Expensive;
•••• = Very complex. €€€€ = Very expensive.

2.1. FUSED DEPOSITION MODELING (FDM)

2.1.1. Process, Materials, Application Areas

Home printers typically work with plastic filament. The technology behind this is often referred to Fused Deposition Modeling (FDM) is a 3D printing technology that works by extruding a thermoplastic polymer through a heated nozzle which gets deposited on a building stage. FDM is also considered to be a form of additive manufacturing, which at the same time is a "process of joining materials to make objects from 3D model data, usually layer upon layer".

Creating a 3D printed object through FDM requires, in the first place, to work on a STL file (stereo lithography file format) which mathematically slices and orients the model for the next building process. Sometimes, the software is capable of generating support structures for the object automatically. In general, the machine requires materials for both the object and the support. [11]

The mere process involves a plastic filament which is fed by a spool to the nozzle where the material is liquefied and "drawn" on the platform. As soon as it touches the build stage, the filament hardens while being gradually deposited, following a certain structure, in order to create the final 3D print. When a layer is drawn, the platform lowers by one layer thickness so that the printer is able to start working on the next layer.
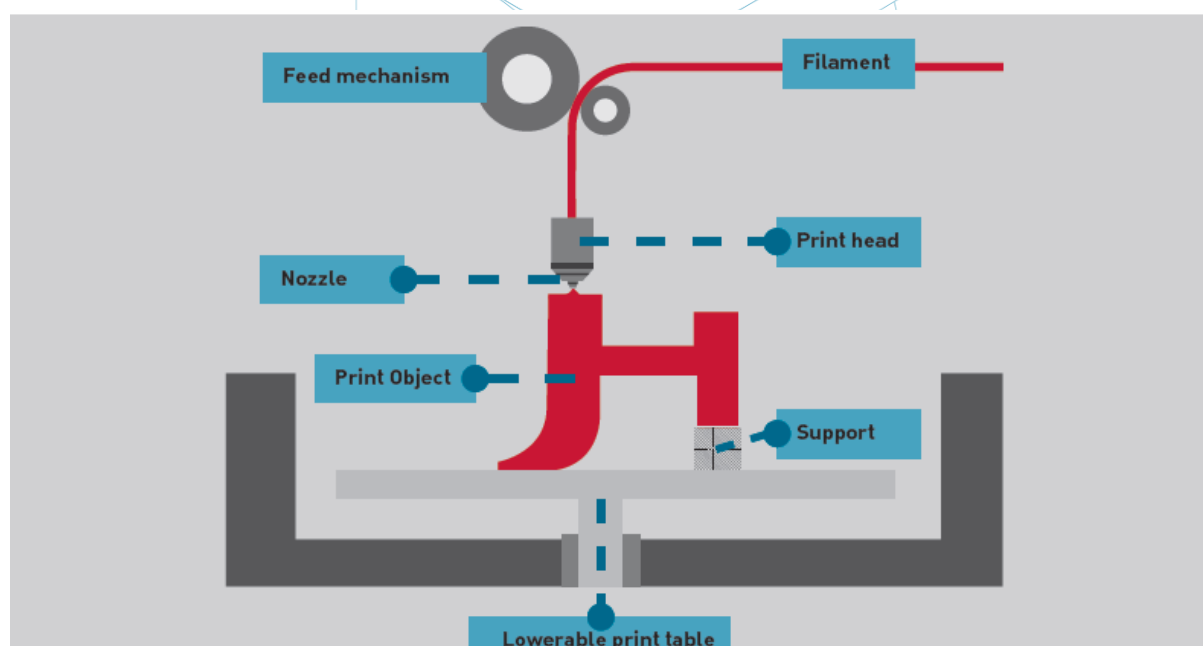
Figure 7: FDM Technology [12]

There are many different materials which can be used with FDM. In the first place, they are divided between the industrial and the consumer categories. The most commonly used are ABS (Acrylonitrile Butadiene Styrene), PLA (Polyactic Acid) and Nylon (Polyamide), but other exotic varieties of materials can also be used, like a material blend of plastic and wood or carbon. [13]

Because this technology presents some very good pros, FDM is often used in the area of non-functional prototypes in order to produce concept parts, functional models, prototypes in general, manufacturing tooling and modeling, and end use parts. More specifically, FDM can be used for low-volume production and prototypes aimed at form, fit and function tests.

At the same time, it is most commonly used in the aerospace sector, for example, to produce wind turbines. Anatomical models for medical use are also very much suitable to be built with this technology. Finally, FDM has slowly been enabling the rapid prototyping of biomedical micro devices, the kind of devices that are used on a daily basis in hospitals, for example, therefore very much fundamental, as it is considered both cheap, but at the same very safe. [14]

Since 2004, FDM technology has been used in a particular sector to produce load-bearing scaffold, which, according to a study, "has the potential for osteochondral defect repair". [15]

2.1.2. Strengths and Weaknesses

When it comes to 3D printing technology, one of the very first concerns relate to its cost. While in general it is the long-term use of materials that can become a serious expense, those who want to engage with Fused Deposition Modeling have an advantage from the start; in fact, FDM printing machines are among the cheapest and most affordable especially for those who want to use it in a domestic environment. Many brands today are available pre-built like Maker Bot and Ultimaker, two of the most popular desktop 3D printers, or can be built using DIY kits or going from the grounds up and printing parts to create a 3D printer.

Always on a positive side, FDM is considered to be a very clean technology, usually simple-to-use and office-friendly. The technology can also produce complex geometries and cavities that would otherwise be quite problematic. [16]

As for accuracy, the 3D prints do not reach the same level of accuracy and quality of other items which are instead produced through the use of Stereolithograpy. That said, the result is considered to be fairly qualitative, depending on the sector where the technology is applied.

Resolution depends mainly on the size of the nozzle that is used. The precision of the machine is dependent on the extruder movements on the X and Y axis, but there are other factors to be taken into consideration. For example, the bonding force between the layers is lower than in the Stereolithography process. Consequently, the weight of the layers might squeeze the lower layers, which can therefore influence and even compromise the quality of the 3D print. Contrary to SLA, FDM presents also an increased complexity. One needs to keep in mind weight and size, but also constraints. It is very important to make sure that a print can meet the expectations that one sees on the screen when first modelling. The constraints in this case depend on several factors, but mainly on the material chosen, through which it is possible to understand how big an item can be printed through FDM. [17]

Another negative side of FDM is that it presents a generally slower compared to both Sterelithography and Selective Laser Sintering, which is to be analysed next. Moreover, Surface finish is considered fair, but not at the same level as these other two technologies. In fact, when printing with FDM, one needs to take into the consideration that the chances are a print will require post processing and finishing.

Choose FDM When You Need...

> High accuracy
> Functional parts
> Durable parts with high stability
> Production-grade materials
> Fast lead times

Technical Specifications for FDM

Standard lead time	Minimum of 4 working days (or 48 hours for models using the Fast Lanes service), depending on part size, number of components and finishing degrees
Standard accuracy	± 0.15% (with lower limit on ± 0.2 mm)
Minimum wall thickness	1 mm
Layer thickness	0.18 – 0.25 mm (varies depending on the chosen material)
Maximum part dimensions	Dimensions are unlimited as components may be composed of several sub-parts. The maximum build envelope is 914 x 610 x 914 mm
Surface structure	Unfinished parts typically have a rough surface but all kinds of fine finishes are possible. FDM parts can be sandblasted, smoothed, colored/impregnated, painted and coated

Figure 8: FDM advantages. [18]

2.2. SELECTIVE LASER SINTERING (SLS)

2.2.1. Process, Materials, Application Areas

Selective Laser Sintering (SLS) is a technique that uses laser as power source to form solid 3D objects. This technique was developed by Carl Deckard, a student of Texas University, and his professor Joe Beaman in 1980s. Later on they took part in foundation of Desk Top Manufacturing (DTM) Corp., that was sold to its big competitor 3D Systems in 2001. SLS in some way is very similar to Selective Laser Sintering. The main difference between SLS and SLA is that it uses powdered material in the vat instead of liquid resin.

Unlike Stereolithography and Fused Deposition Modeling, Selective Laser Sintering does not require the use of support structures, therefore cutting the amount of materials that one needs to provide for the print. The object is, in fact, printed while being constantly surrounded by not sintered powder.

The mere process involves a laser which is used to selectively sinter a layer of granules, consequently binding together the material to create a solid form. At the end of the process, the object can very hot and therefore it will be left to cool off before being removed from the machine.
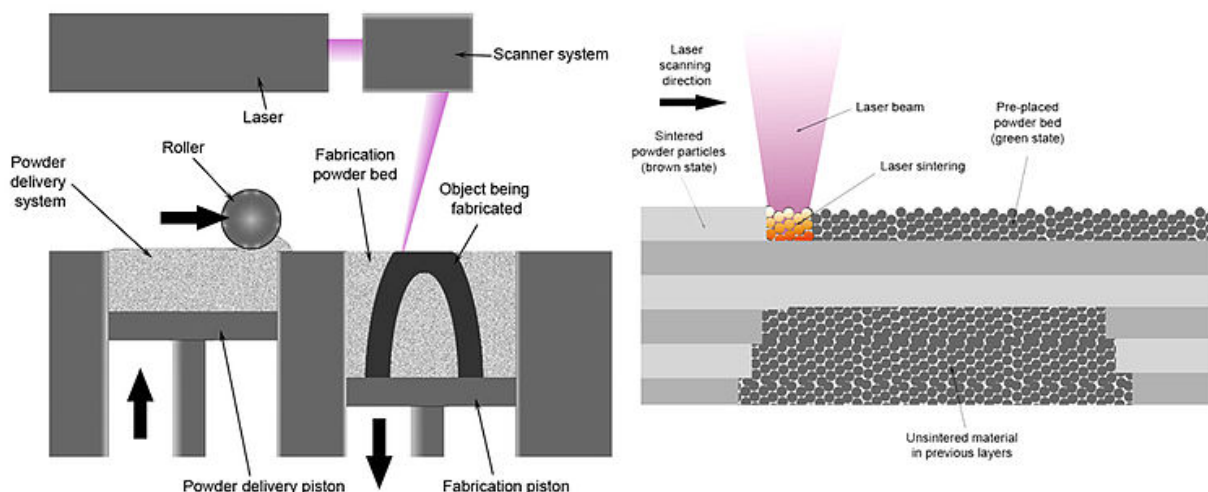
Figure 9: SLS technology. [19]

The use of SLS technology makes it possible to involve a variety of materials which range from nylon, glass and ceramics, to aluminium, silver and even steel. However, some of them, like ceramics, are not laser sintered. A binder, in this case, is used to glue parts together and this is usually known as "Powder & Binder-based 3D Printing".

The starting process of this technology is quite similar to Laser Sintering: a roller puts a thin layer of powder on a platform. However, instead of a laser beam, a special print head places a binding agent at specific points, printing a thin layer of your model that is able to bind to subsequent layers. This process is then repeated over and over again until your model is complete.

Since in this case the model was only "glued" together, some post processing is. The exact post-processing steps heavily depend on the material: Multicolour models get a bath in superglue, Ceramic prints are put in a drying oven and fired several times, High Detail Stainless Steel and Steel objects are placed in an oven for fusing. Steel models are additionally infused with Bronze for extra strength. [20]

Areas of application of SLS technology range from the automotive sector to the consumer goods sector. More specifically, it can be used in the case of product development and rapid prototyping in a wide range of commercial industries, as well as limited-run manufacturing of end-use parts. In the aerospace industry, for example, SLS is involved to build prototypes for aircraft components. This represents a great advantage for companies because airplanes are produced in small quantities since airlines companies, for example, use them for quite a long time. Therefore, it is not cost-effective for production companies to build physical moulds for airplane parts. In fact, these moulds would be too expensive to make and would then need to be stored for long periods of time without being damaged or corroded. [21] Therefore, SLS is perfect for small batches production.

Even more interesting is the fact that SLS has been slowly introduced in the sector of consumer goods, specifically for production of shoes. According the producers, the use of nylon sintered by SLS can produce a shoe with not only reduced weight, but also customizations, while being able to adapt the shoe to the needs and the requirements of the customer. [22]

Figure 10: Shoe silhouette created using SLS. [23]

Success with titanium would be one example of huge future opportunities with the military, aerospace, medical and other industries looking to utilize titanium's light weight, strength and corrosion resistance. Much of the ground-breaking activity in 3D printing has utilized plastics and moving from plastics into metals of all types is the next step.

2.2.2. Strengths and Weaknesses

One of the very first pros that Selective Laser Sintering presents is the fact that it makes no use of support structures, as it is fully self-supporting. It therefore allows parts to be built within other parts in a process called nesting. This has two main outcomes; the first one is that it reduces the costs of the material which would therefore be used in order to provide support structures, greatly used in the FDM technology. The second advantage in this case is that SLS can handle a high complexity of geometry. Some products are that complex that without this technology it would be complicated to produce them.

Generally, SLS is considered the 3D printing technology with the fastest additive manufacturing process for printing functional, durable prototypes and end user parts. Durability is, moreover, supported by the use of strong materials like nylon, which also allows a certain freedom in the functionality of the final 3D print. Besides, thanks to its excellent mechanical properties, the material used in SLS is often a substitute of typical injection moulding plastics.

At the same time, SLS produces parts which are generally rated to be very strong and stiff, with good chemical resistance. Complex parts with interior components, channels, can be built without trapping the material inside and altering the surface from support removal.

Accuracy is another important advantage of SLS technology. Final 3D prints present, usually, a high accuracy. The required process is also fast, compared to the already analysed technologies. At the same time, scalability allows to use SLS for a single part of component, but also and easily for dozens of production pieces. Parts can be usually shipped between 1 and 4 days, which is a great advantage for companies who are required to be quick in the market.

However, usually SLS prints present a certain surface porosity hence why, just like in Fused Deposition Modeling, post-processing is definitely required.

2.3. STEREOLITHOGRAPHY (SLA)

2.3.1. Process, Materials, Application Areas

Stereolithography (SLA) is a light-based process that builds individual layers of a model with liquid polymer, hardened by a laser beam. It is the oldest technology in the history of 3D printing, but it is still very much used nowadays.

The process involves the use of a printing machine called "stereolithograph apparatus" which converts liquid plastic into solid 3D objects. The laser is directed and controlled by two galvanometers. After each layer, the resin tank peels away to release the hardened material. The building platform then moves up from 25 to 200 microns, depending on the chosen layer height, to prepare for the process of solidifying the next layer. The part appears to be built upside down, which is called inverse stereolithography.

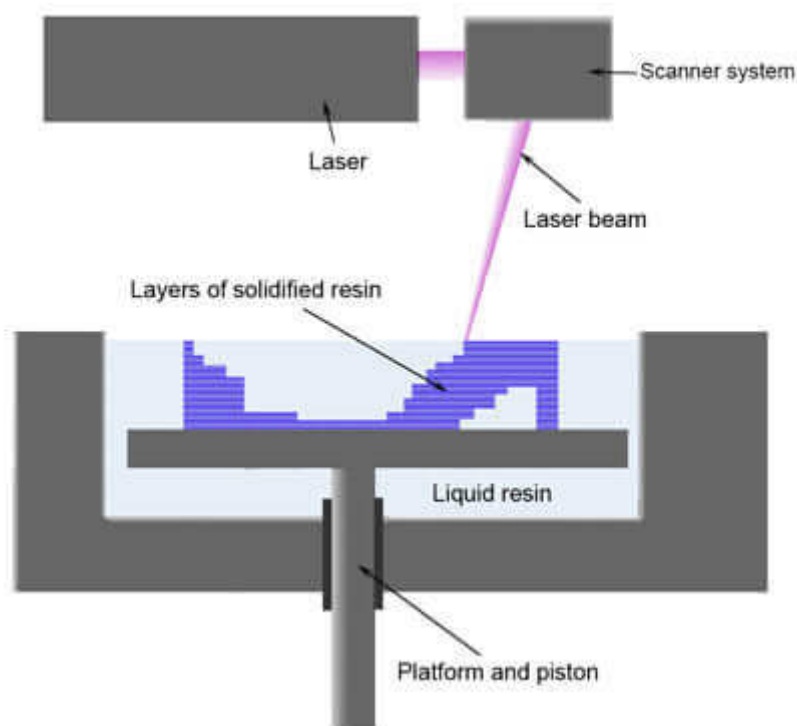*Figure 11: Resin 3D Printer parts overview. [24]*

When it comes to materials, Stereolithography offers a variety of choices which can be summarized in three categories. Within the first, paintable resin can be used. In this case, the process takes place in a large tank, and begins when a layer of liquid polymer is spread over a platform. A computer-controlled laser draws the first layer onto the surface of a liquid

polymer, which hardens. As the model is then lowered, the next layer is drawn directly on top of the previous one and repeated until the model is finished. When the model is complete, it is raised out of the tank and the excess liquid flows away. The supports are removed manually after the model is taken from the machine.

Transparent resin and prime grey are two other kinds of resin material that can be used as well. From a simple magnifier glass to waveguide to a see-through mechanism or virtually unlimited decorative objects, transparent resin can deliver amazing results. Just as with translucent resins, if one adds some lights to their project printed with this material one will definitely grab people's attention.

This material can be mixed with pigments before printing resulting in an attractive looking transparent colour object. The transparency varies from translucent to water-clear. The water clear transparency can be achieved in special cases and upon request. Transparent resin has a great surface quality. The surface is smooth, and the stair-stepping look of 3D printing can be reduced by sandpapering. Transparent resin models are painted with a shiny varnish to prevent discolouration from UV light.

The natural surface of your transparent resin model is known as basic finish. Resin based models will always be smoother than powder-based models. The technology used for your transparent resin model requires extra material to support your model during the printing process, but these supports are removed before your model is cured.

The surface of the 3D printed model is defined by the material and technology. With basic finish, some building layers will still be visible. The natural surface option is always the cheapest. The number of post-finishing steps also determines the cost and quality of your transparent resin model.

The material is transparent, but not 100% water clear. When a model is solid with a thickness greater than 2 cm, the printed part presents a bluish tint. For the transparent finish, the model is painted with colourless varnish. For transparent colour finishes, pigment is added to the varnish.

Grey Resin instead, formerly known as Prime Grey, is suitable for A-side visual models with limited functionality. The surface of the material is very smooth, much smoother in fact than almost all other 3D printing materials. The colour is Air Force Grey and the material feels almost "luxurious" to the touch. The material has a medium mechanical resistance. Freedom of design is limited because of the structure necessary to support your models during printing.

Models made in Grey Resin are typically used as high-quality show models. Design and engineering departments use the models as visual prototypes or for presentations, but we have noticed that it works really well for character and toy models.

The difference between transparent resin and prime grey stays in the result that one wants to accomplish. For example, Prime Grey is excellent when it comes to its impact strength and the accuracy in the details. At the same time, paintable resin allows to choose a colour at one's choice, but it still needs support structure, therefore limiting the freedom of design. This can be avoided using prime grey. A summary table for all three kinds of resins is available below.

Name	Impact strength	Stiffness	Humidity resistance	Heat resistance	Durability	Appearance	Mould Making	Details	Description	Applications
Transparent resin	Good	Good	Excellent	Sensitive	Good	Optical clear with a light blue tinge	Excellent	Moderate	Tough Impact resistant High elongation at break Excellent surface quality	Functional prototypes Wind tunnel testing Water flow analysis High-end finished models ABS-like parts
Paintable resin	Good	Good	Good	Good	Good	White	Excellent	Good	Tough Good surface quality Good thermal properties Durable	Impellers Duct work and connectors Automotive housings Dashboard assemblies High-end finished models
Prime Gray	Excellent	Good	Good	Good	Good	Gray	Good	Excellent	Tough Impact resistant High elongation at break Excellent surface quality	Tough enclosures Snap-fit assemblies Replacing CNC machined parts High-end finished models

Figure 12: Resins, properties. [25]

Stereolithography can be used in many areas. Nowadays, the automotive, aerospace, medical and consumer goods fields are the most popular ones. It can, in fact, be applied to the domestic field: one can easily make 3D prints starting from their own model. At the same time, this technology allows to print medical models of anatomic human parts, which can be very much useful in class, for example, in order to give a proper overview on the functionality of the human body to a group of scholars. It therefore changes the way subjects are taught and it can be applied in technical subjects like science, engineering, but also arts and math.

2.3.2. Strengths and Weaknesses

Among all the 3D printing technologies, Stereolithography is one of the best; its strength relies mainly on the high resolution of the 3D prints. It allows to print objects with very complex geometries while at the same time preserving the quality and the details. Accuracy is in fact generally rated to be very good in this case. SLA is all about precision and it is often used where form, fit and assembly are critical. This is especially due to the use of the resin, in particular the so-called photopolymer materials.

The latter are, in fact, liquid materials which are cured and hardened; they also allow a certain freedom when it comes to colours, opacity and rigidity, while offering an excellent surface quality. At the same time, although there are faster and more recent technologies out there, Stereolithography can perform correctly in a reasonable period of time, while also allowing to save time on highly precise parts. In this way, prototypes, which is a common use for Stereolithography, can be made easily and faithfully to the initial design. Complexity is, therefore, not a problem.

Moreover, the positive side of using Stereolithography and resins material is customization, especially in the form of which colour one wants. Spray print in this case is one of the methods used to achieve this result. Four kinds of paints are available: dead matt, matt, satin, and high gloss, each with a different gloss factor. The higher the gloss factor, the shinier the model will look. Paint sprayed on larger areas will exhibit bigger visual differences between the different paint options. We can spray paint, your model to match your needs or you could also do this at home yourself.

However, costs are one of the major weaknesses in the case of Stereolithography. While 3D printing machines can be more or less affordable, photopolymer materials can be very expensive, making the ordinary use of this technology not affordable to everyone. Besides, although they come in different colours, there is still a limited choice of photopolymers.

Cons when it comes to Stereolithography involve also the fact that liquid resins are generally irritating and toxic, therefore some precautions are to be taken in order to work with them using the appropriate tools and equipment. Moreover, prints usually require cleaning, which can in some cases take time and a lot of effort. Post-processing is generally needed if one wants to achieve the highest quality possible. This will also depend on the material chosen, which will consequently lead to more or less steps to be taken care of.

Not to be forgotten is the fact that Stereolithography, contrary to its similar technology which is Selective Laser Sintering, requires support structures, which makes the price more expensive, as more material is required. The distinction and the consequent choice between these two technologies
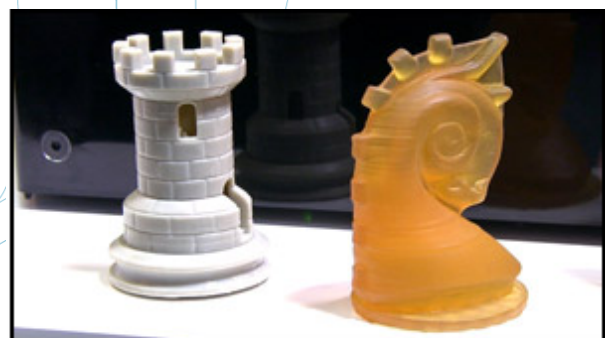
Figure 13: Photocentric 3D prints. [26]

mainly depends on the material one wants to use and the result one wants to achieve.

3. PRODUCTION PROCESS IN 3D PRINTING

Hereunder, in this chapter, all the necessary processes and steps for, starting with a digital design, obtaining a real printed piece, are going to be described.

It is important to mention that there is not just a single valid process for printing three dimensional pieces. What is explained in this guide are a certain number of steps that should be adapted to the type of piece, selected technology, type of machine, and even to the used software. Furthermore, the process that is described hereunder is mostly intended for fused deposition modelling (FDM) 3D printers.

The production process, in general, is the following:

1. Obtaining the digital model

2. Exporting and repairing the STL file

3. Testing, orientation, distribution and G-Code

4. 3D printing

5. Extracting pieces

6. Post-processing
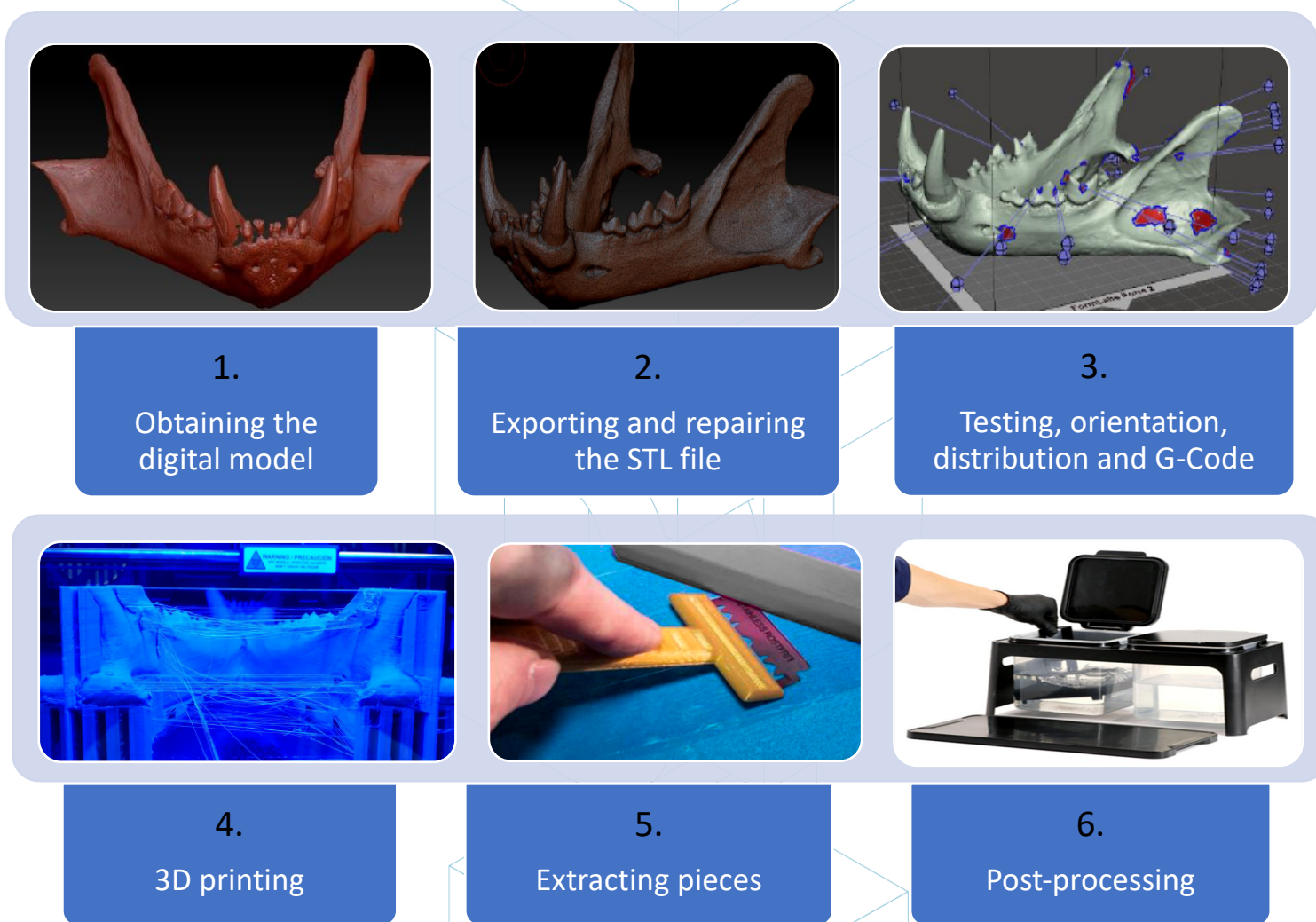
Scheme 2: Production process in 3D printing. [27]

It is important that the tips given in each of the steps of the process are indicative and they have general term. This means that the tips should not be followed very closely. In the production process for 3D printing, experience, piece features, used machine, etc. have a lot of weight. It is for sure that someone with little or without experience will 3D print a lot of piece with failures, before he finds the key.

Now, each of these steps of the production process are explained in more detail and deeper.

3.1. Obtaining the digital model

There are several possibilities for obtaining the 3D model or digital model that is wanted to print. In particular, there are three possibilities:

- **To model the piece using a CAD software:** With this first option, in order to get the three-dimensional model, a computer aided design (CAD) software must be used. There are a lot of available CAD software for modelling, and there is not a best option; it will depend on the user and their abilities with the software.
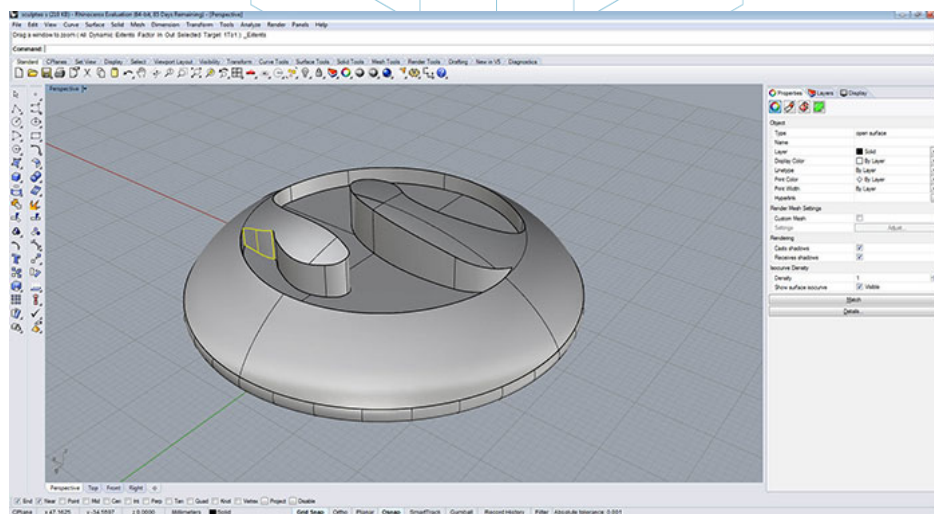
Figure 14: Piece modelled with Rhinoceros. [28]

- **To obtain the geometry by Reverse Engineering and 3D scanners:** By this option a 3D scanner is used to digitally obtain the geometry of a real object. This is not a simple process and some ability and experience is required. In addition, there are several types of 3D scanners, and they are usually expensive.

  The reverse engineering process usually is for copying, improving, or customizing real objects, or also for incorporating complex surfaces to a 3D modelled piece.

The process is normally the following: firstly, the geometry is captured by a 3D scanner. Then, this geometry is treated or modified, so that it is adequate for 3D printing. That is to say, if it is obtained a point cloud from the 3D scanning, this cloud has to be "sewed" and converted to a polygonal mesh (sometimes the 3D scanner software does it) and finally exported (converted) to a solid body. Finally, the process of 3D printing will be implemented.
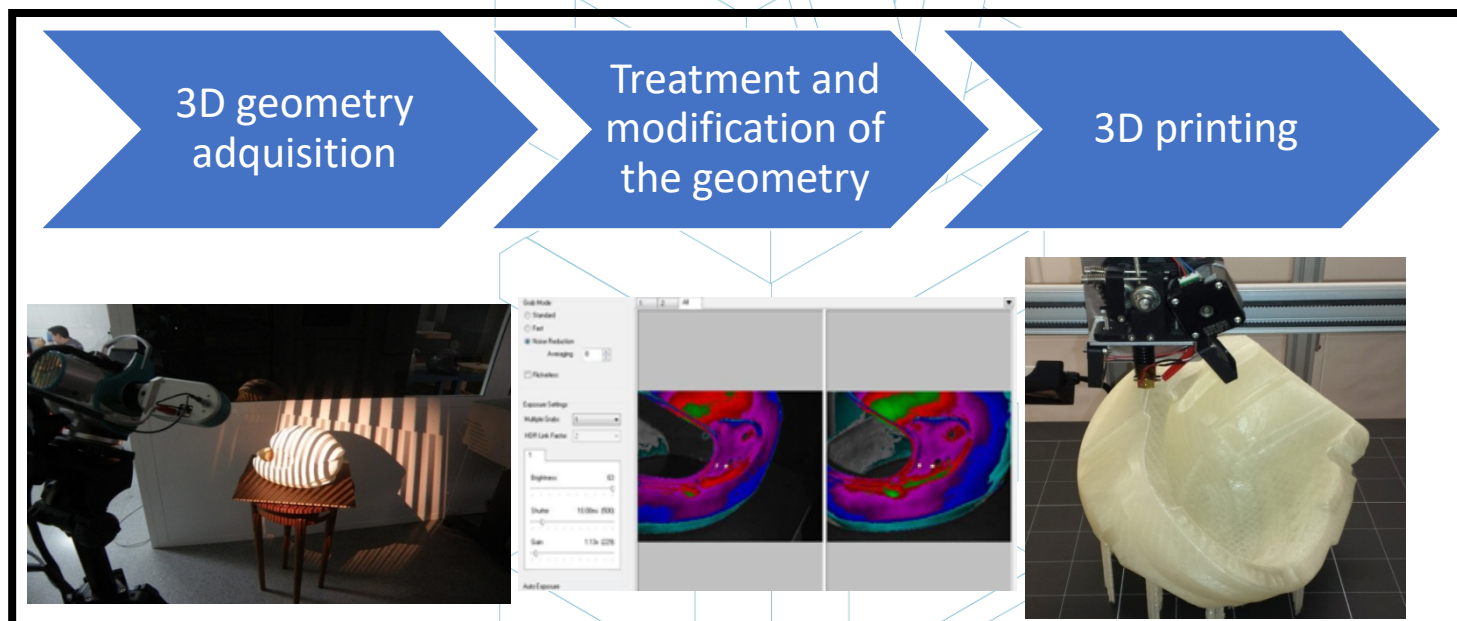
Figure 15: Reverse engineering process. [29]

- **Downloading the model from repositories or asking someone to design it for you:** If you do not have knowledge in computer-aided 3D design, or you do not have the necessary equipment (or software, or even knowledge) for applying a process of reverse engineering, to download the model from a repository or to ask somebody to design it for you is the best option. Depending on whether the repository is a 3D printing model repository (e.g.: Thingiverse) or a more general digital model repository (e.g.: GrabCAD), the downloaded model will be ready for 3D printing or it will be not.
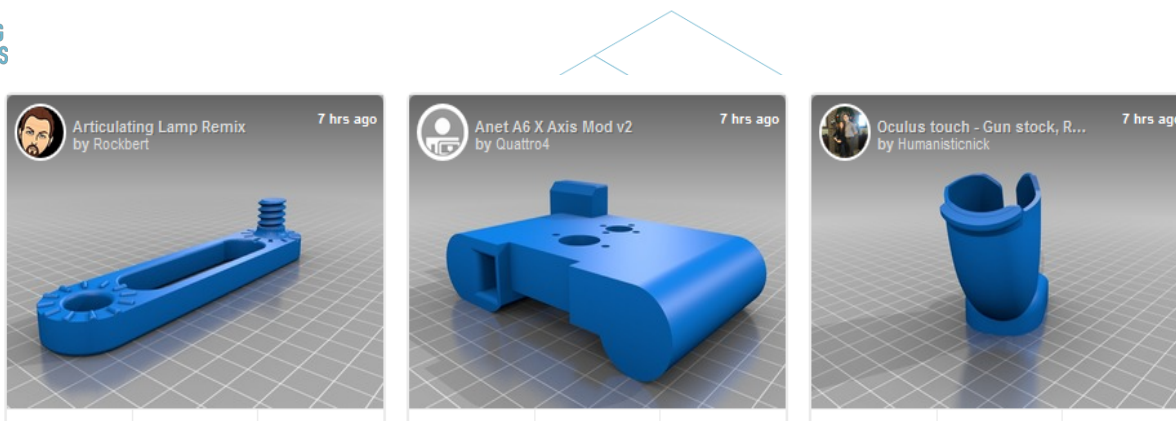
Figure 16: Examples of pieces from Thingiverse repository. [30]

No matter how the digital model is obtained, what it is important is to get a robust 3D model, thought for 3D printing. That is to say that any designed and modelled piece, or scanned piece, or downloaded from the internet, could not be adequate for being 3D printed. This means that some issues such as internal gaps, the precision to achieve, the detail level, the size of the piece, if the piece has overhangs, make it a closed solid (watertight), etc. should be taken into account.

In the next tutorial it can be seen a range of CAD design tips for 3D printing, depending on the material selected: https://www.sculpteo.com/en/materials/materials-design-guidelines/

And in the next link it can be found more information and more tutorials about how modelling and preparing a piece for being 3D printed by different CAD software: https://www.sculpteo.com/en/tutorial/

## 3.2.   Exporting and repairing the STL file

When you work designing and printing 3D models, a wide range of formats or types of files are available. Some of them are thought for designing or scanning, but others are associated to 3D printing, such as: STL, OBJ, PLY or FBX, among others. Depending on the modelled piece, on the software, on the 3D printer features, etc., one or another format must be used. In this guide, in order to unify criteria, it is explained how to export and use the STL file.

When the piece is designed and modelled, a format conversion to the ". stl" file is needed. If the piece has been downloaded from a repository, this conversion is frequently already done. However, if free or commercial CAD software has been used this conversion is needed.

First of all, what is STL format and how it works is explained briefly:

STL stands for "Standard Triangle Language". In simple terms, this format uses linked triangles for recreating the solid model surface. Depending on the model complexity, more or less

triangles (and with more or less size) will be necessary for recreating it, and the greater or the less big the size of the file also will be. In the file, each triangle is defined by a set of parameters, such us the normal vector to the triangle face, and the coordinates (x, y, z) of each triangle vertex.
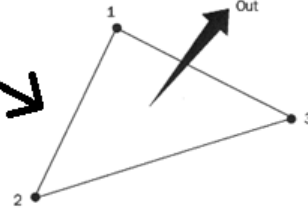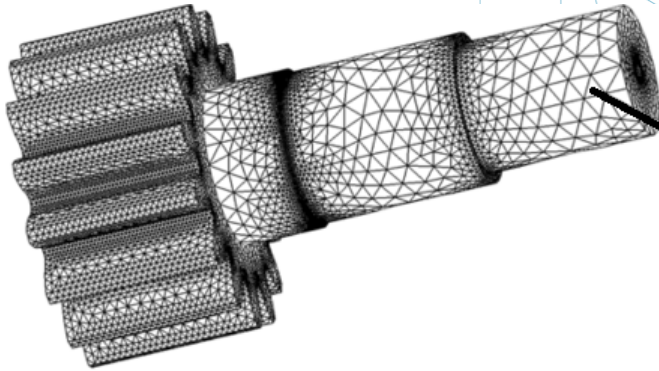

```
facet normal n_i n_j n_k
    outer loop
        vertex v1_x v1_y v1_z
        vertex v2_x v2_y v2_z
        vertex v3_x v3_y v3_z
    endloop
endfacet
```

Figure 17: How STL file works. [31]

Normally, exporting a CAD design to STL format is as simple as going to the used software menu and clicking on "Save as..." or in "Export" and choosing STL. Also, depending on the software, some characteristics such as precision or tolerance must be chosen. Hereunder the steps to follow in some CAD programs are shown:

Software used	Step 1	Step 2	Step 3	Step 4	Step 5
Catia	1. Select STL Command	2. Maximum Sag = .0003" or .0125 mm	3. Select part(s) to be converted and Click YES	4. Select Export	5. Type filename and output the STL
Inventor	1. Select: Save Copy As	2. Select: STL	3. Select Options Menu: Set To High	4. Enter Filename	5. Save
ProEngineer	1. File / Export / Model	2. Choose STL	3. Chord Height: .0005" or .0125 mm	4. Angle Control: .5	5. Click: APPLY
Rhino	1. FILE / SAVE AS	2. Select File Type: STL	3. Enter a File Name and Save	4. Select Binary File	
SolidEdge	1. FILE / SAVE AS	2. Set Save as Type: STL then select Options	3. Set: Conversion Tolerance: .0005" or .0125mm	4. Set Surface Plane to: 45.00 (degrees)	5. Save
SolidWorks	1. FILE / SAVE AS	2. SAVE AS TYPE / Select: STL	3. Select:Options	4. Deviation Tolerance: .0004" and Angle Tolerance: 7.75 deg	5. Save
Unigraphics	1. FILE / EXPORT / Rapid Prototyping	2. Triangle Tolerance: .0005" or .0125 mm	3. Adjacency Tolerance: 0.12	4. Click: APPLY	5. Set Auto Normal Gen to: ON, Normal Display to: OFF, Triangle Display to: ON

Figure 18: Exporting to STL in different software. [32]

Sometimes, there are problems during conversion to STL, either because the model is not thought for 3D printing, either because the design in the CAD software has not being made correctly, or either other causes. So, the exported model may have some errors.

These errors are of various kinds: holes or gaps, reversed triangles (it means, with the normal vectors in opposite direction), duplicated faces or triangles, faces or triangles that intersect, singular points or faces (out of the model), etc.

The digital model reparation is explained in the next step of the production process in 3D printing, because it is very linked to the implementation of analysis to the pieces.

3.3. Testing, orientation, distribution and G-Code

This stage of the production process in 3D printing is about the preparation of the pieces or digital models (previously exported to STL) for being 3D printed. It is about performing the next process, orderly:

- To analyze the piece or model: thickness, holes, stability, angles, triangle mesh, etc.
- To design the support structures or to automatically generate them.
- To choose the infill of the model, both percentage and shape.
- To locate the piece on the printing surface (or printing bed) and to select the most appropriate orientation.
- To generate the machine code or G-Code.

3.3.1. Analyzing the piece or model

The analysis usually is necessary when pieces are relatively complex, or when the origin of the pieces is not known, or it can be done just if you want to be one hundred per cent that the piece is suitable for being 3D printing. In addition, a good analysis can detect errors in the triangle mesh arisen from the STL conversion.

These analyses can be implemented by some software, that also is useful for other purposes or not. Implemented analysis of:

- *Thickness*: Recommended thickness depends on the 3D printing machine (and frequently on the used technology). Some machines allow to bigger thickness than others. To search the specific machine and to see the allowed thickness should be enough. Generally speaking, for fused deposition modelling (FDM) machines, thickness about 1 millimetre should be considered, approximately. This parameter must not be confused with layer thickness. The parameter here referred is like the thickness of the object itself. This analysis is needed to be done if it is thought that the model needs it.
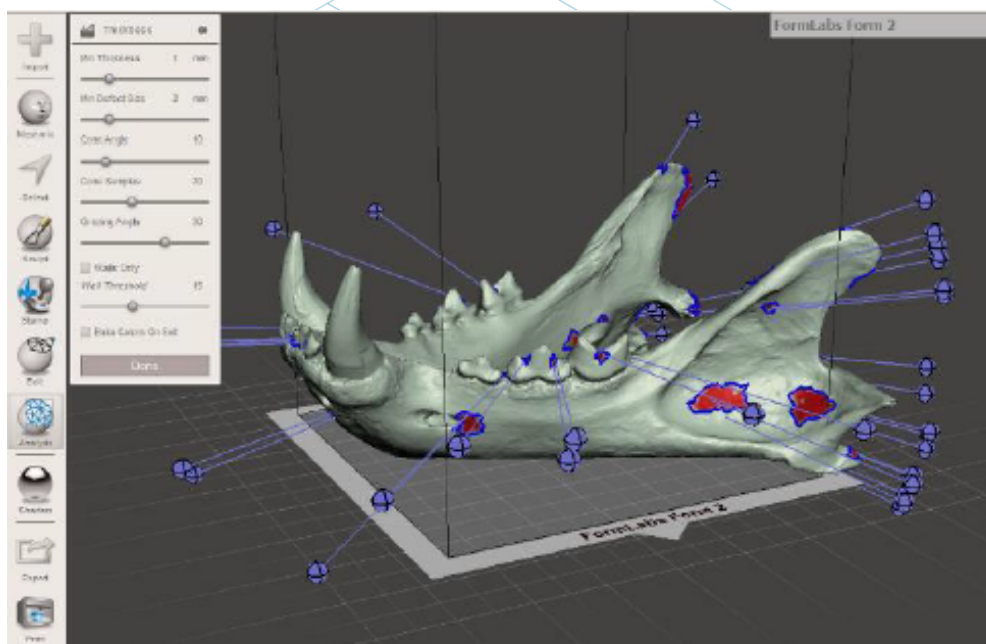
Figure 19: Thickness analysis (1 millimetre) implemented with Meshmixer. [33]

- *Holes or gaps*: The model want to be printed must be perfectly closed, or best said: it must be watertight. This means that the triangular mesh must not have holes or gaps, or what is the same, it must not have vertex or triangle points not joined; every of them must be connected to other triangles. It is highly recommended to perform this test always, before printing, because it can be errors or failures hard to be seen.

- *Angles and overhang*: By this analysis it can be found, depending on the selected technology and machine, if the model or piece will need of support structures for being printed. Generally, for FDM printers the minimum allowed inclination angle is 45 degrees. Parts with greater angles will require of support structures. This analysis sometimes is not needed, because lots of pre-printing programs directly calculate the needed supports for the pieces.
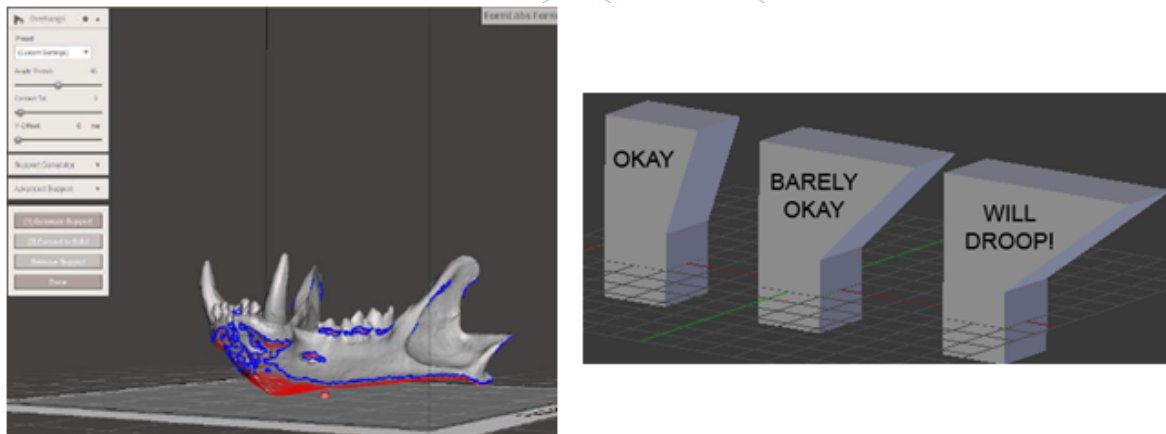
Figure 20: Angle analysis. [34]

‒ *Others*: Depending on the used software, more or less analysis will be able to be done. Also, by certain programs, standard analysis can be done, by them, different kinds of problems may arise.

Finally, it is noticeable that many of the used programs for analysing the piece allows, not just to detect errors or problems, but also allow to repair, or what is best, auto repair the model.

3.3.2. Support structures

For some technologies (almost all the ones that use plastic), it is necessary that, in order to beat gravity and to print overhang parts (or with internal gaps), support structures are inserted on these zones. They are usually necessary from 45 degrees (for FDM printers).
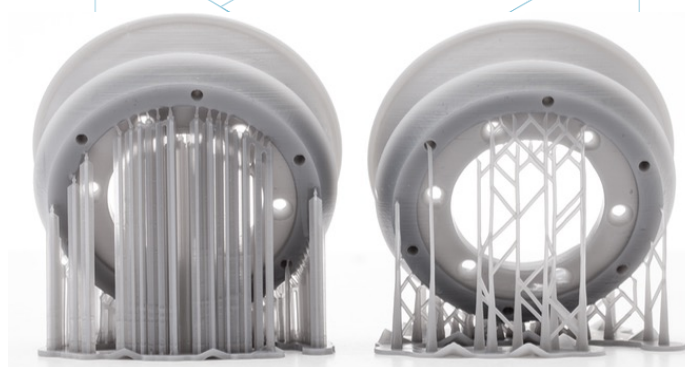

Figure 21: Support structures. [35]

Support structures are usually done by the same material as the piece, although there are 3D printers that print two materials: piece's one and support structures one. With these printers soluble support materials in certain liquids can be used.

As support structures are thought just for holding the first layers of the model that are cantilevered or "floating", support structures are built lightly and using less material than for the piece itself. In addition, they will not mark so much the piece, when they are removed.

Most of the available software, either analysis software or either the own software of the printer machine, allow two options: making a design of the support structures, or automatically calculating and inserting these structures. If you do not have enough experience in 3D printing it is recommended let the program automatically calculates the supports. Although it has to be taken into account that it may not optimize the used material. In addition, a single software may have several different designs for support structures.

A good guide for designing, using and calculate how and when using supports can be found in the following link: https://www.3dhubs.com/knowledge-base/supports-3d-printing-technology-overview. In addition, this guide is adapted to different technologies (FDM, SLA, PolyJet, SLS...).
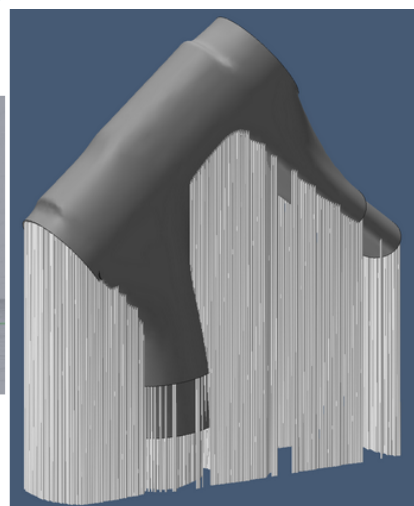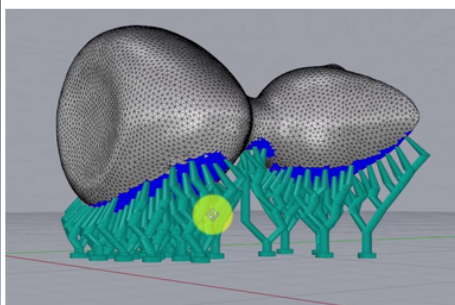

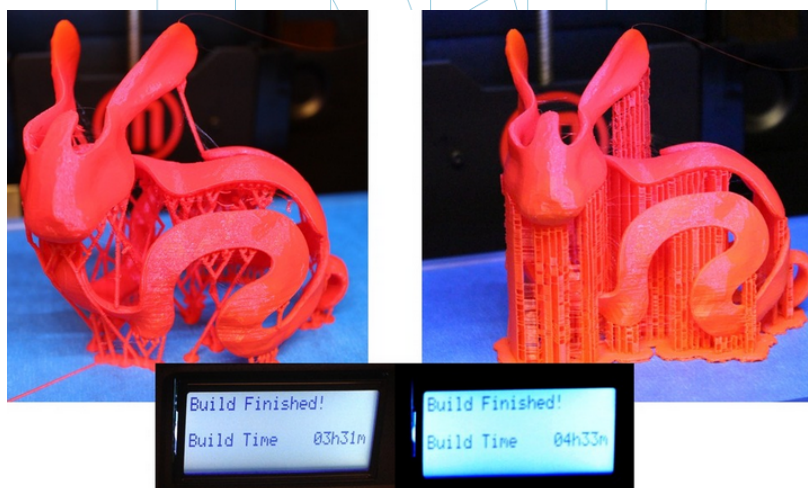*Figure 22: Different types of support structures. [36]*

Figure 23: Differences between a piece with optimized support structures (left) and non-optimized (right). Differences between printing times can also be seen. [37]

3.3.3. Model infill

This step can be implemented before inserting the support structures.

When we talk about the infill it is about the structure printed inside the object. It means, if we think in the example of a cube, the six external walls will be printed in a solid way, with a certain thickness, but the internal part of the cube will not be solid; you will have to choose the infill percentage, and even the geometrical shape of the infill. With some visual examples it can be better understood:
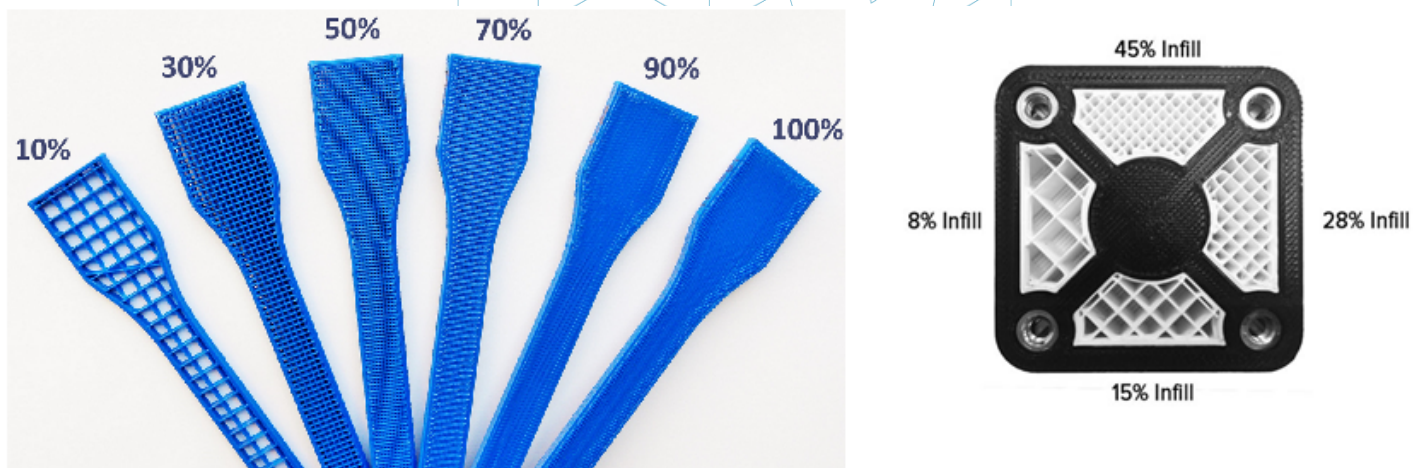

Figure 24: Different infill percentages. [38]

In the figure above are seen two examples of pieces with different infill percentages. Percentage will be chosen (in the pre-printing software) depending on if the piece must have more or less resistance, according to weight.

The geometrical pattern of the infill can be also chosen. Some of them are more resistant than others, but generally, the default pattern of the software can be selected.

Percentage and pattern shape will be chosen according to several aspects: total weight of the piece, used material, resistance to be achieved, printing time and sometimes decorative features. In general, the greater the infill percentage, the stronger the printed piece will be, but the longer will take to be printed. A percentage about 15% usually is enough.

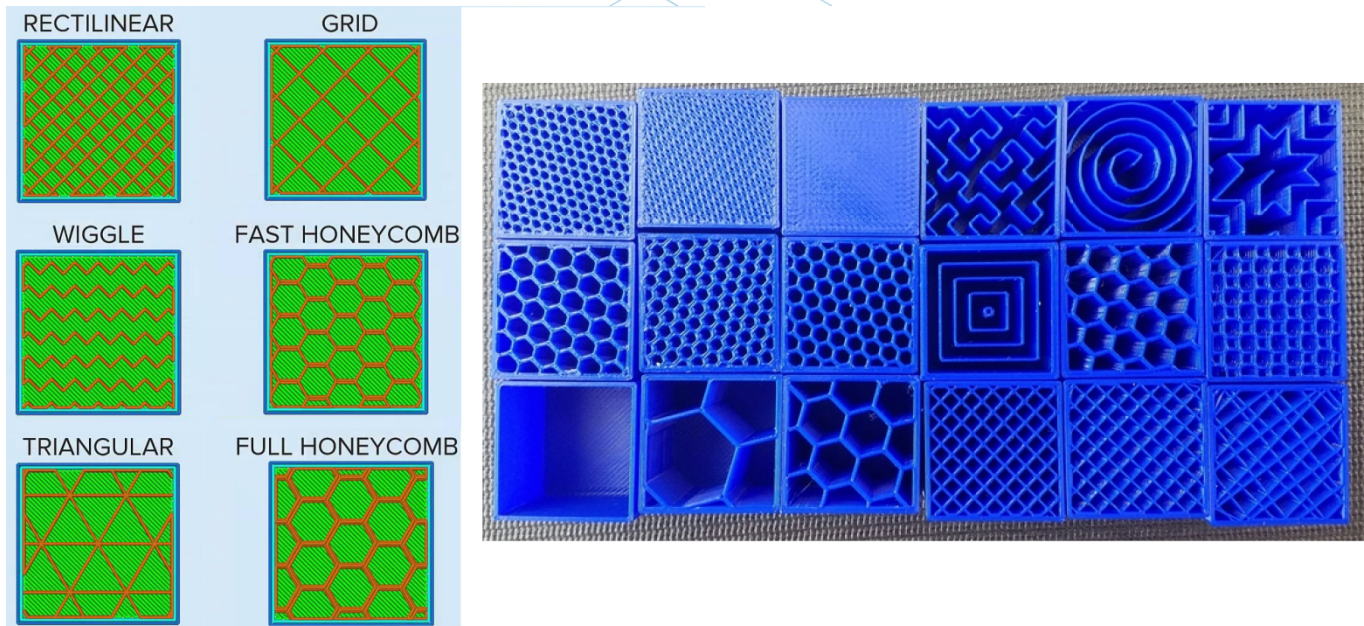Hereunder some examples can be seen (on software and on printed pieces):

Figure 25: Different infill patterns. [39]

3.3.4. Positioning and orientation

To decide the position and orientation of the piece on the printing surface or printing bed is one of the most important part of the whole process. It is a decision that will have a great impact on the piece quality and properties.

It is necessary to take into account that one positioning or another will be done depending on the technology used to print (FDM, SLA, etc.). In this guide it will be explained mostly according to fused filament printers (FDM)

One of the most used criteria for choosing position and orientation is using the minimum amount of material (and spending less printing time). This is achieved by minimizing overhang parts. Consequently, less support structures will be printed, and the piece will be built in less time. However, sometimes quality to achieve is more important, so that orientations that are not optimal in terms of material and printing time, can be chosen.

Some tips, in a general way, are:

- To center the pieces in the printing surface or bed surface. This will reduce the movements of the printing head (and, consequently, the printing time). Furthermore, it will increase the quality and precision of the piece, because printing platforms are usually more leveled and calibrated in its central part, and also because if they are heated, heat is greater in the central part.

- If you want to print several pieces at once, they should be put together, with a gap between them of about 5 - 15 millimeters.

- If there are curved or sloped surfaces, and these parts of the piece are desired to be printed with quality, piece should be positioned so that these surfaces are positioned on XY plane (horizontal plane), or as parallel as possible to this plane. So "stair-stepping" effect will be avoided, in which curved or sloped surfaces are not smooth.
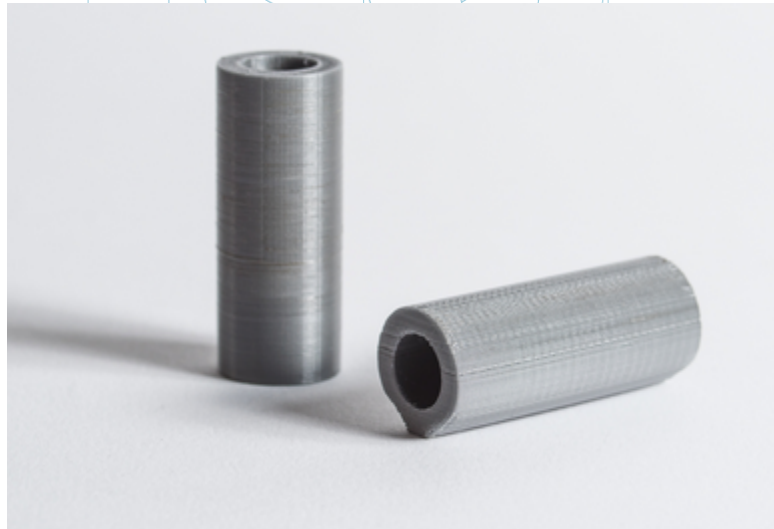

Figure 26: Quality differences in the same piece, according orientation. [40]

- Watching the previous picture, if the piece has an inner hole, or a through hole, it would be adequate to put this hole with its axis perpendicular to the printing bed, if a great quality is required in the hole surface.

- A very long and plane section printed on the horizontal plane or XY plane may suffer deformation, which means that its outer borders get cold and shrink very quick, causing the piece to warp upwards. Sometimes, to print these pieces so their longest section is perpendicular to the building plate is convenient.

- In a general way, the top surface of a printed piece will have the best finishing.

- If we are printing functional pieces, that have to withstand forces and loads, it is much more likely for them to de-laminate and to break when the forces or loads are perpendicular to the layers direction. This can graphically be seen in the next picture:
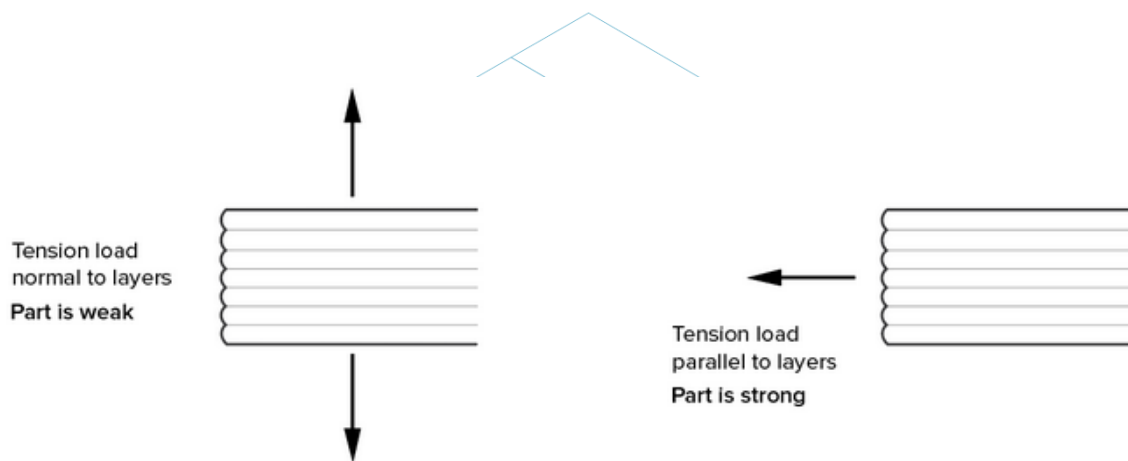
Tension load
normal to layers

Part is weak

Tension load
parallel to layers

Part is strong
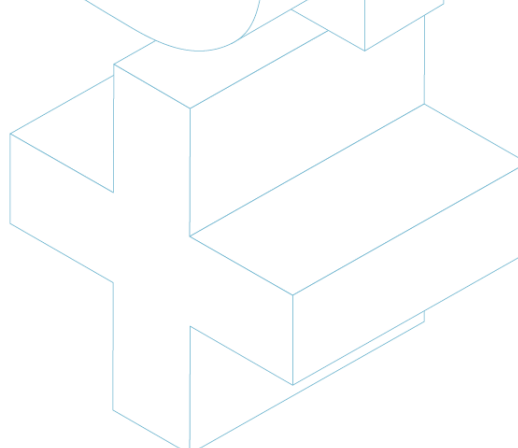
Figure 27: Tension load. [41]

3.3.5. Generating the G-Code.

Once all the previous steps have being developed, it is time to generate what is known as G-Code or machine code. This code is the translation of the piece (and all the parameters that have been previously set) into instructions that the machine can understand.

Before that, depending on the used software, it is necessary to choose the **layer height or layer thickness.** This parameter has also a lot of importance and it will have a great impact on the final quality of the surface of the piece. A higher layer height will result on a great resolution or quality, but also will result on a greater printing time.

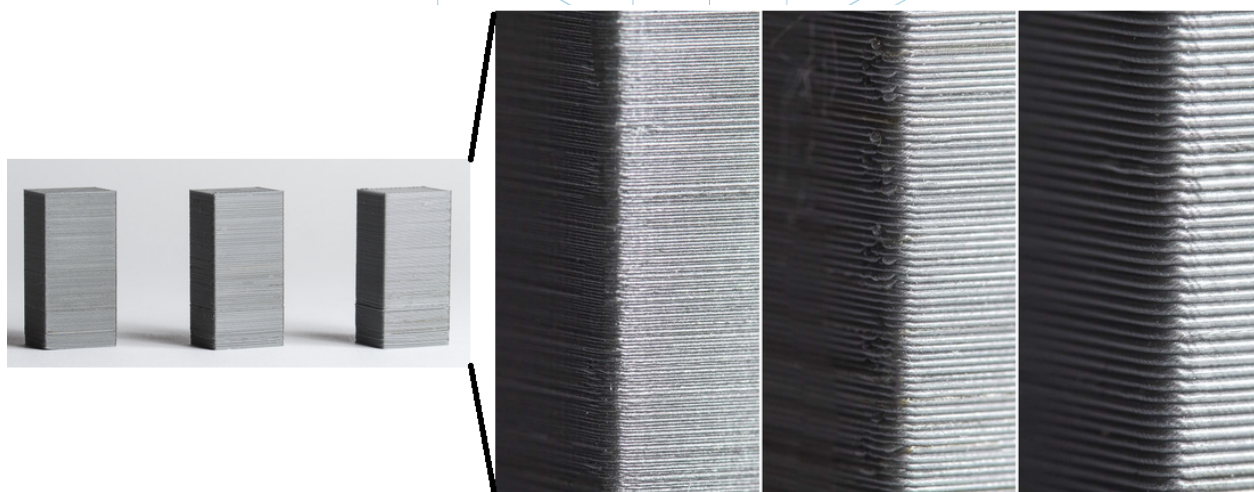*Figure 28: Different layers thickness. [42]*

It is quite important to understand what is more important: aesthetics or a quick and cheap printing. Sometimes, differences between two identical pieces, but with layers height of 100 microns and 200 microns are hard to distinguish. But the 100 microns piece will take the double to print and will cost more. Therefore, it is very important to know what the final use

of a piece is. It is also crucial to know the number of curves and angles of the piece, because the layers height is more visible on these parts than on straight walls.

Figure 29: Two different layer heights. [43]

To sum up with the layer height matter, it is noticeable that for each machine and technology a certain range of layer height values is available to be chosen. For example, for FDM, a very common range is from 50 to 300 microns (0.05 to 0.3 millimeters).

So, at this point the G-Code must be generated, and this can be done by various software. Probably, some of the programs to be used may have already be used in a previous step. The program cuts the model into horizontal "slices" or layers, generating the path that the machine head will follow. It also calculates the amount of material (weight and meters) and the estimated printing time.
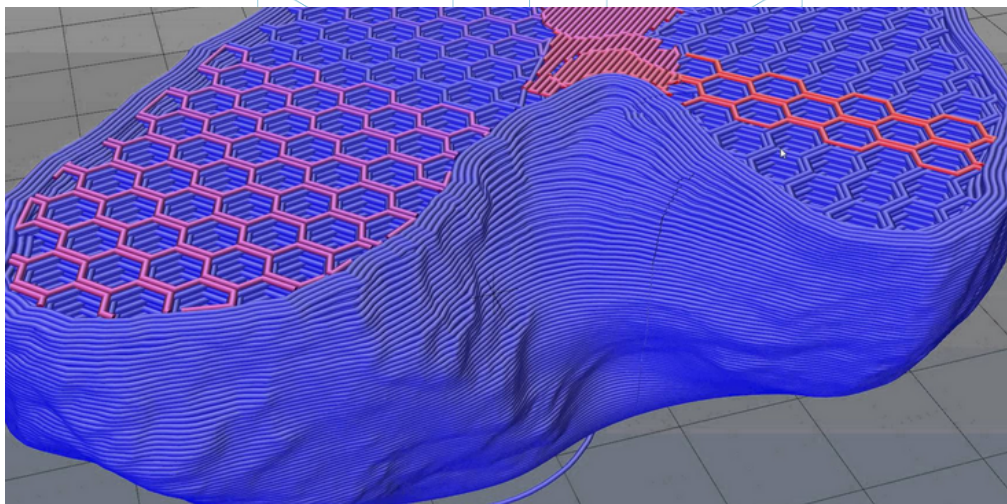

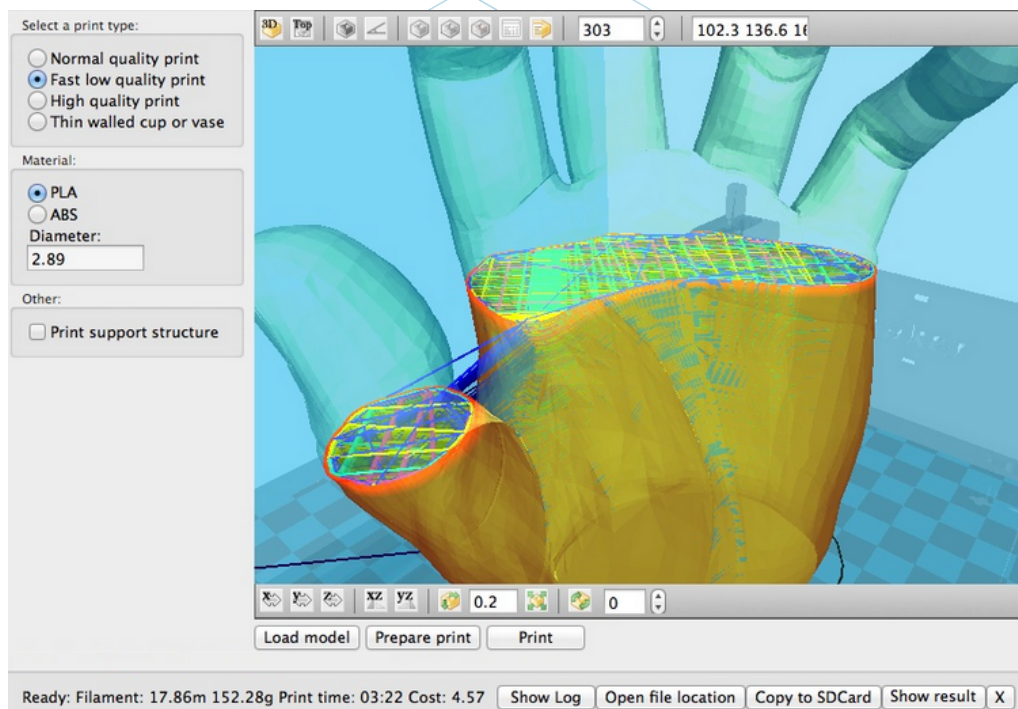*Figure 30: Detail of the layers of a piece. [44]*

With the chosen program, the G-Code is exported and saved, in a SD card or in a pen drive. Or it can be also directly sent to the machine, depending on the software and machine.

To sum up this part, it should be taken into account that there are a lot of parameters that can be changed: wall thickness, printing speeds, printing temperatures and bed temperature, flow, etc. But these are parameters to edit by users with experience, and it is advisable to let them default.

3.4.   3D printing

Now the G-Code has being obtained the process of 3D printing can be implemented. There are a few things to consider and to check before printing.

For FDM printers, the temperatures of the bed or printing platform and of the extruder must to be checked (the software or the printer do that automatically when you start to print). To use some kind of lacquer or varnish is also recommendable in order to ease the extraction of the piece. It is advisable to read the instructions to also know how to load the filament on the printer for example, and for further information.

Therefore, the G-Code is sent to the machine, and the machine starts to work.

3.5. Extracting pieces

When the printer has already finished, it is the moment to implement the process of extracting the piece/s. Once again, depending on the machine and mainly, on the technology and used materials, the extraction processes will vary.

For fused deposition modelling machines (FDM), pieces often are removed by hand, or with some tool like a blade. It is also possible to have flexible platforms that ease a lot the extraction. There are even other methods such as using a solvent or applying cold or with dental floss.

Figure 32: Extraction. [46]

For further information about extraction processes for FDM printers, the following link is given: https://all3dp.com/1/remove-3d-print-from-bed-stuck-glass/

For other technologies, extraction processes changes. Those technologies that use liquid plastic buckets and lasers (for example), or those that use powder materials, have different extraction processes:
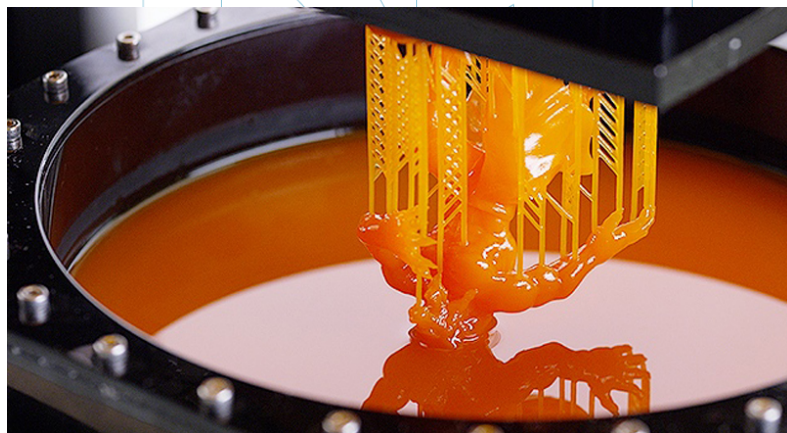

Figure 33: SLA extraction process. [47]

Figure 34: SLS extraction process. [48]

3.6.   Post-processing

For a lot of pieces, a finishing process is needed, that once again varies along the kind of technology and kind of machine. For some technologies and machines this process is necessary.

Since there are a lot of technologies and machines, here there will exposed the most common processes, and also the most used for FDM printers.

First of all, it is important to note that some of the printed pieces may not need a post-processing step. Or maybe some of them will need just to remove mechanically the support structures.
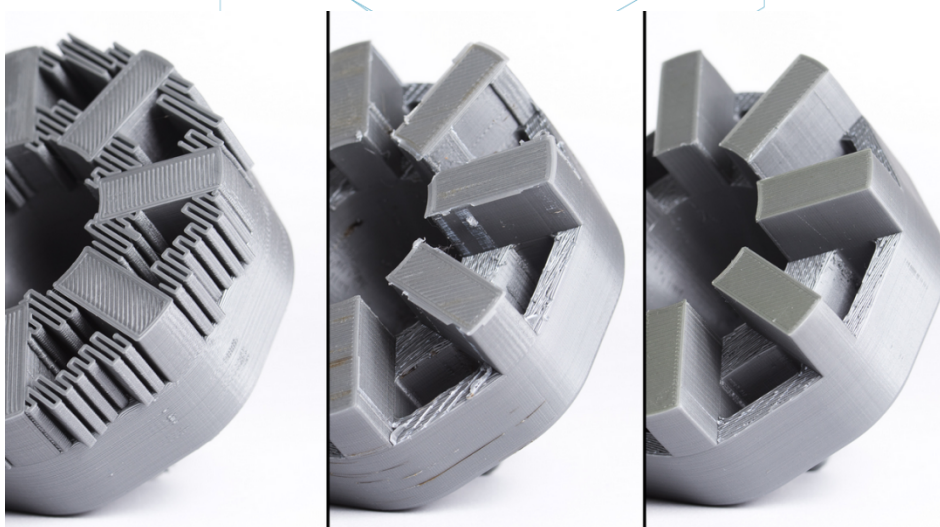

Figure 35: Finishing process. [49]

There are a lot of finishing processes and post-processes:

- *Remove support structures:* This can be done mechanically, as it has been said, or by the use of a solvent bath (even water), if the printing material is the adequate one.

Figure 36: Mechanical process. [50]

Figure 37: Bath for removing the support structures. [51]

- *Sanding:* It is a process to apply when the support structures have been already removed. Several sanding grades can be chosen.
- *Polishing:* If the piece is wanted to have a "mirror-like" surface it is necessary to polish the piece. It is needed to sand the piece with a sandpaper of number 2000, previously. Then, the powder should be cleaned, and the piece will be polished with a microfiber cloth and a special enamel, which will produce a lasting shine. There are also rotary sanders and polishers.

- *Coatings:* All the pieces are capable of applying a coating. In the case of paints, it is advisable to apply first a coat layer and then to paint with aerosol, acrylics or airbrushes. Epoxy gels, metal coatings, etc. are also available.
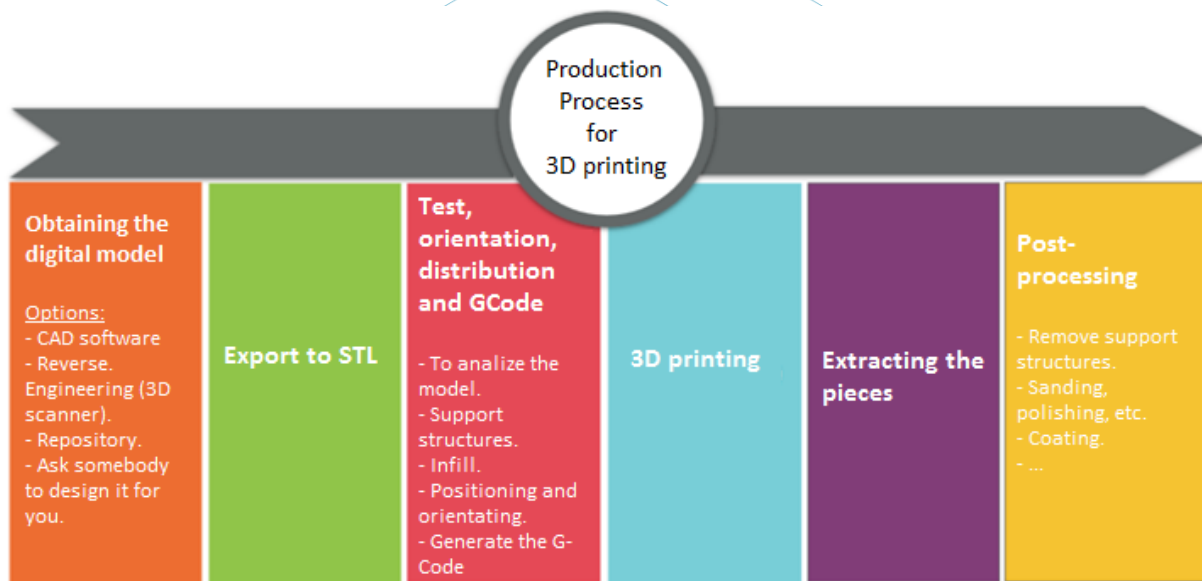

Figure 38: Different finishing. [52]

There are more finishing processes: shot blasting, smoothing with acetone vapor... For knowing more post-processes for FDM technology the following link is provided: https://www.3dhubs.com/knowledge-base/post-processing-fdm-printed-parts

- *Curing oven:* For other technologies, such as stereolithography, sometimes it is necessary some curing post-process in an ultraviolet rays oven. This process is implemented in order to improve the properties of the piece's material, among others.

For more post-processes or finishing processes for other technologies, the following links are provided:

- For SLS: https://www.3dhubs.com/knowledge-base/post-processing-sls-printed-parts
- For SLA: https://www.3dhubs.com/knowledge-base/post-processing-sla-printed-parts
- For FDM and PolyJet: https://www.stratasysdirect.com/wp-content/uploads/2016/10/finishing-for-fdm-and-polyjet-Rebrandeds-2.pdf

The whole process can be collected in the next scheme:

Scheme 3: Production process for 3D printing. [53]

4. SOFTWARE THAT YOU NEED FOR 3D PRINTING

There are different software for each form of creation of the 3D object, as well as for each phase of the process. To know which software you have the most affinity and meets your needs, you have to study and evaluate which one is most appropriate for your workflow and your technical level.

The process for obtaining the 3D object is composed of 3 phases: Designing the 3D object, Repairing/Generating of the G-Code and Printing.
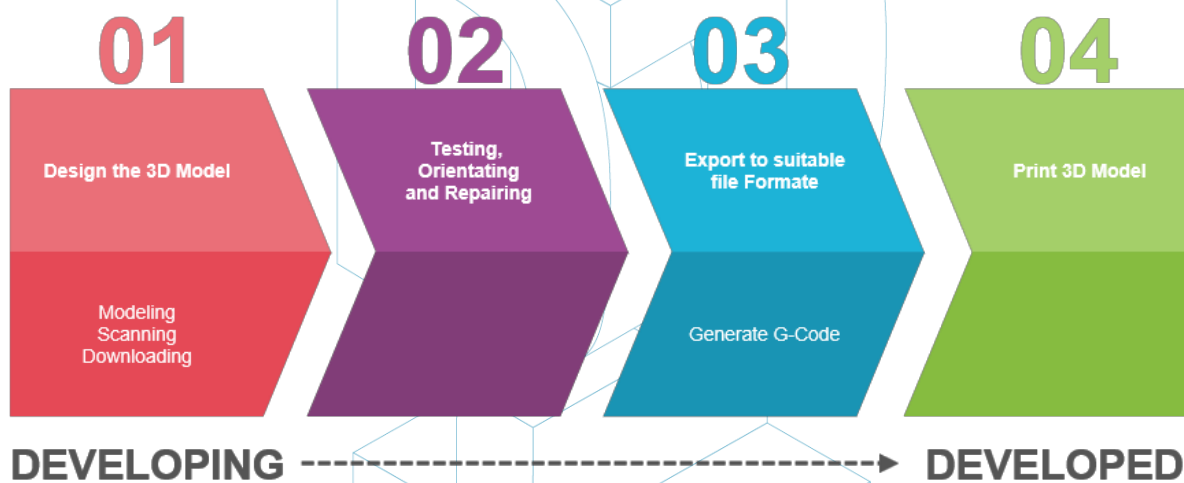

Figure 39: Process to develop a 3D Model. [54]

Some software are capable of doing the whole process, which means they have tools for 3D object creation, verification and repair, until the generation of G-Code.

Below we explain the software available for creation of 3D object; they are divided into 3 groups.

4.1. Programmes for designing

There are three ways to create the 3D object: Modelling, Scanning and Downloading repositories that are already available online.

Modelling

Many programmes can be used to create or draw a 3D Model since the beginning. They have different levels of complexity, also different licenses. To choose which of them meets your work needs and fits to your abilities, we list them below, with a brief description and links, that you can you to know more about them.

Free software

SketchUp	Its powerful yet easy to use interface make it ideal for beginners in 3D modeling.	https://www.youtube.com/watch?v=pv7Tr...
blender	Open source and free software for 3D design. It is also used for animation, rendering, video editing	https://www.blender.org/support/tutorials...
FreeCAD *An Open Source parametric 3D CAD modeler*	Parametric 3D modeler. It is easy to modify designs using the model history and changing the parameters. The program is a multi-platform (Windows, Mac OS, and Linux) tool, and reads and writes various open file formats	https://www.freecadweb.org/wiki/Draft_t...
WINGS 3D	Advanced subdivision modeler that is powerful and easy to use. It is open source and completely free.	http://www.wings3d.com/?page_id=252

Comercial sofware – Student version

AUTOCAD	CAD and drafting software application for 2D and 3D design	https://knowledge.autodesk.com/support/...
AUTODESK INVENTOR	Professional 3D CAD software offers an easy-to-use set of tools for 3D mechanical design, documentation, and product simulation and digital prototyping	https://www.youtube.com/watch?v=lEheF...
AUTODESK REVIT	Revit is specifically built for Building Information Modeling (BIM), empowering design and construction professionals to bring ideas from concept to construction with a coordinated and consistent model-based approach	https://www.youtube.com/watch?v=PR08...

Comercial software

SOLIDWORKS	It is a 3D design solution for rapid creation of parts, assemblies, and 2D drawings. Application-specific tools for sheet metal, weldments, surfacing, and mold tool and the make it easy to deliver best-in-class designs	https://www.youtube.com/watch?v=LAadl...pW_KMa
CATIA	Leading software solution used to design, simulate, analyze, and manufacture products in a variety of industries	https://www.youtube.com/watch?v=gGbq...
CINEMA 4D by MAXON	3D modeling, animation, motion graphic and rendering application capable of procedural and polygonal/subdivision modeling, animating, lighting, texturing, rendering, and common features found in 3D modelling applications	https://www.youtube.com/watch?v=iLQzB...

Table 1: Software to Designing a 3D Model. [55]

Scanning:

By using 3D scanning you will be able to capture a digital copy of a physical real-world object. Different technologies are able to scan an object to create a 3D Model, as example: time-of-flight, structured / modulated light, volumetric scanning etc. Some scanners are listed here:

STRUCTURE SENSOR	https://structure.io/
Sense™ 3D Scanner	https://www.3dsystems.com/3d-scanners/sense-scanner
XYZ PRINTING	http://eu.xyzprinting.com/eu_en/Product/da-Vinci-1.0-AiO#view
KINECT	https://developer.microsoft.com/en-us/windows/hardware/3d-print/scanning-with-kinect

Table 2: Software to Scanning a 3D Model. [56]

Downloading

The third possibility to have your 3D Model is downloading it from websites of repositories, read to print or that needs a small correction. By clicking in the link below, you can find many websites, where different examples of Models are available.

http://3dprintingforbeginners.com/3d-model-repositories/

https://www.thingiverse.com/

4.2. Programmes for testing, orientating and repairing

Before you can print your 3D Model, you need to analyse the 3D model. This will show if any part cannot be printed or if there is a failure. Then it is possible to repair, to decrease or increase the triangles that form the surface that will be printed, and to cut the model in parts if necessary.

There are some software available to do this part of the process, below you find a list with the most used:

AUTODESK NETFABB	https://knowledge.autodesk.com/support/netfabb?p=NETF&skill=Beginner&sort=score&page=1&v=2017
Meshlab	http://www.meshlab.net/#support
AUTODESK MESHMIXER	https://www.mmmanual.com/

Table 3: Software to Testing, Orientating and Repair a 3D Model. [57]

Here we use as an example the software Nettfabb, to explain how you can prepare your 3D Model and generate the STL file.
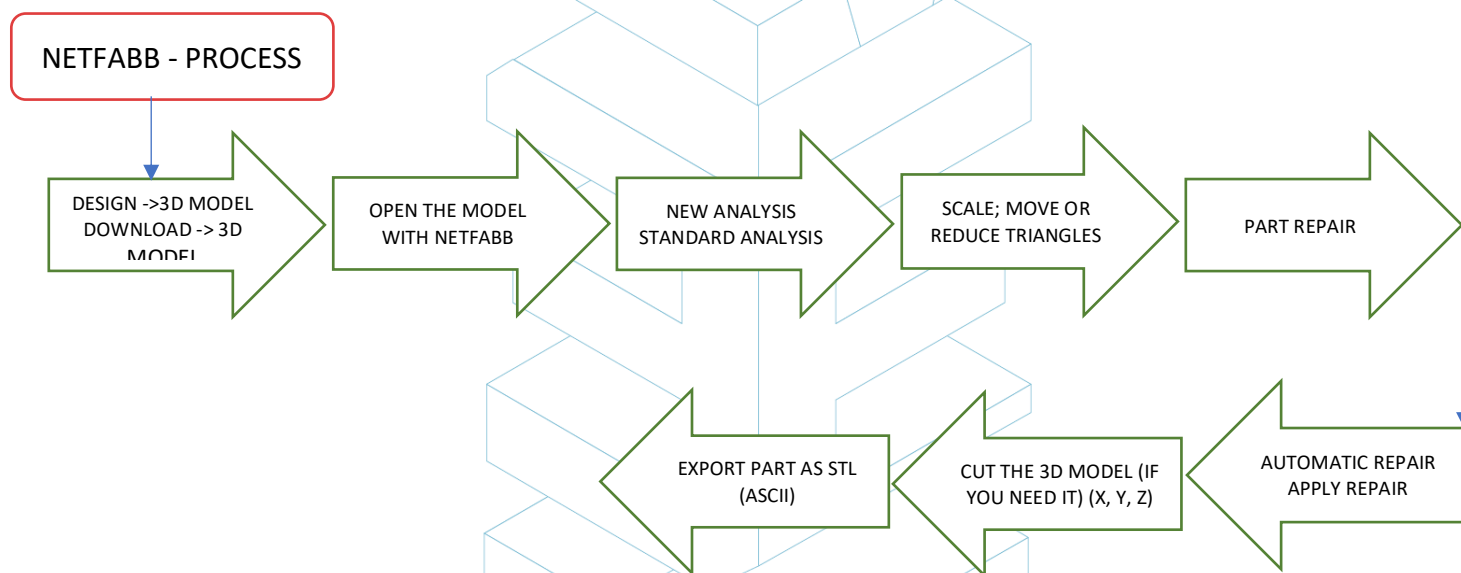
Figure 40: Repairing a STL file. [58]

Below you will find a link with an explanation of how to do the process, as well as the appearance of the program's interface:

Repair it with Netfabb in under 3 Minutes Tutorial:

https://www.netfabb.com/blog/repair-it-netfabb-under-3-minutes
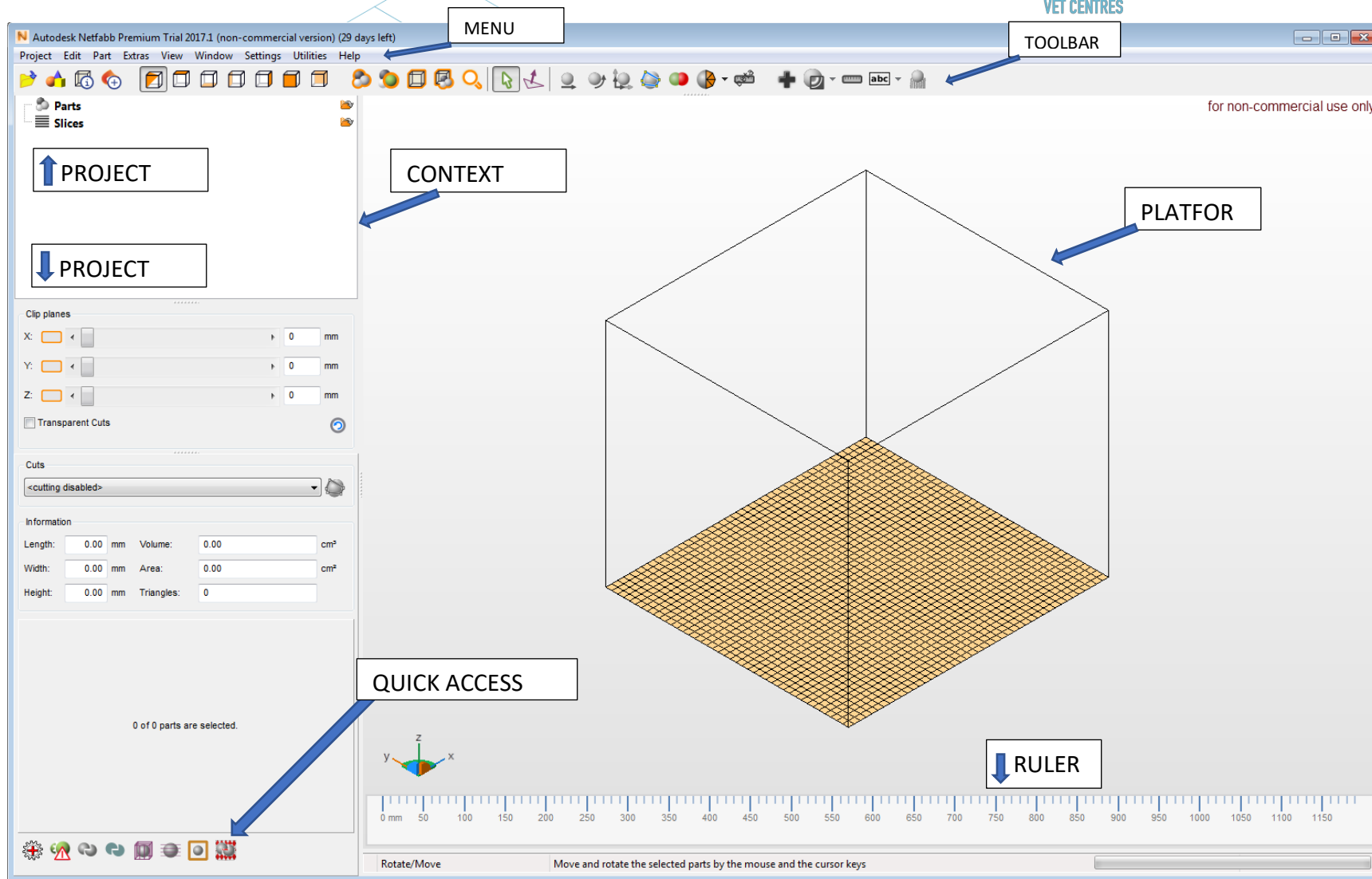
Figure 41: Netfabb Premium Interface. [59]

4.3.   Programs for generating the G-code

G-code is a language in which people tell computerized machine tools how to make something. The "how" is defined by instructions on where to move, how fast to move, and what path to follow. The G-Code can be generated with many different software like Skeinforge, Cura, Slic3r... One of the more used software is Slic3r. It is a necessary tool to convert a 3D model into printing instructions (G-code) for the 3D printer. It cuts the model into horizontal slices (layers), generates toolpaths to fill them and calculates the amount of material to be extruded. It is possible to scale, rotate, split or cut the object, and also to add support.

- Creating the G-Code with Slic3r:

1. Once Slic3r is open, click on the "Add" button, select the model and open it. Slic3r allows to modify, rotate, scale, cut··· the model, but if it is ready to be printed the next step is creating the G-Code. To do it, click on "Export G-Code" and save it.

Figure 42: Open a 3D design with Slic3r. [60]

Figure 43: G-Code with Slic3r. [61]

2. Then the G-Code of the 3D model can be copied in a SD card and inserted in the 3D printer

4.4. 3D Printing Workflow

E3D+VET
ERASMUS+
3D PRINTING
VET CENTRES

REPETIER - HOST

DESIGN

STL

TESTING ORIENTATION REPAIRING

CREATING G-CODE

G-C

G-C

MODELING

SCANNING

DOWNLOADING

NETFABB

MESHLAB

MESHMIXER

SLIC3R

SKENFORGE

KISSLICER

CURA

SUPERSKEIN

MATTERCONTROL

WEB REPOSITORIES:
- THINGIVERSE
- 3D MODEL FREE
- PONOKO
- NASA 3D MODEL
- I.MATERIALISE
- SHAPEWAYS

3D SCANNER:
- KINET (MICROSOFT)
- SENSE 3D SCANNER
- STRUCTURE SENSOR

FREE PROGRAMS:
- SKETCHUP
- WINGS3D
- 123D
- OPENSCAD
- BLENDER
- FREECAD
- OPENSCAD

COMERCIAL PROGRAMS /STUDENT VERSION:
- AUTOCAD
- INVENTOR
- REVIT

COMERCIAL PROGRAMS:
- CATIA
- SOLIDWORKS
- 3D STUDIO
- RHINOCEROS
- CINEMA4D
- ZBRUSH

We have to export our 3D model to the file format more suitable for us,. stl, .obj, etc.

Figure 44: Diagram 3D Printing Workflow.

5. 3D PRINTING MATERIALS

5.1. Overview

When it comes to 3D printing, materials are often one of the most important choice. Earlier AM technologies were making use of certain materials which proved to be non-resistant enough and were degrading quickly. With time, the knowledge about 3D printing has been extending and has been spreading all over the world. With more people interested and the idea that this technology has the right potential to lead to new innovative ways of manufacturing, further studies and analysis has been conducted and new materials have been created.
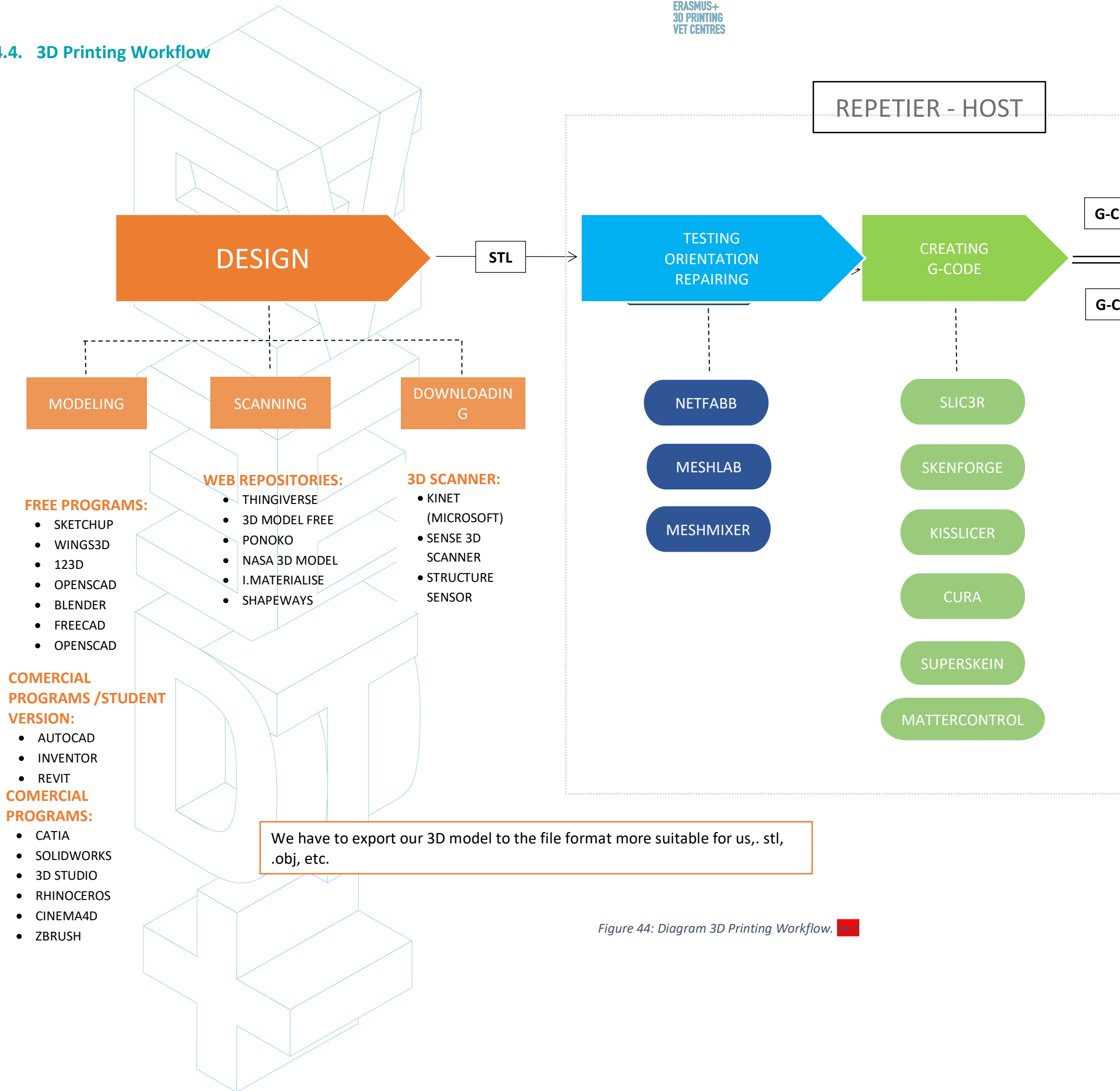
Today, the 3D printing market offers a variety of choice for what concerns materials. From polymers and metals, to ceramics and composites, many are the materials that have been created, each of them with its own advantage and disadvantages. Some examples are visible on 3dhubs.com, a portal which provides 3D printing services on a global level. [63]:

- **Prototyping Plastic**, suitable for fast and cost-effective prototyping;
- **High Detail Resin**, suitable for intricate designs and sculptures;
- **SLS Nylon**, for functional prototypes and end-use parts;
- **Fiber-Reinforced Nylon**, for engineering strong parts;
- **Rigid Opaque Plastic**, for realistic prototypes with high accuracy;
- **Rubber-Like Plastic**, simulating rubber;
- **Transparent Plastic**, to create see-through parts and prototypes;
- **Simulated ABS**, with high precision and functional molds;
- **Full Color Sandstone**, for photo-realistic models;
- **Industrial Metals**, for prototypes and end-use parts.

The industry's offer is, of course, much wider than what is reported here. For what concerns the demand instead, here the use of a certain material is strongly influenced by not just the type of technology adopted, but also the popularity of 3D printing machines.

According to the global 3D printing report for 2016 which collected information from companies using a variety of 3D printing technologies, the demand for materials in this case

shows that metal is leading the competition. Metals are currently being in areas like aerospace and automotive for many reasons. Apart from the high quantity that companies need in this case, 3D printing technology using metals allows to create lightweight components. Weight is, in fact, extremely fundamental for those companies which produce aircraft components, for example. It impacts the quantity of fuel that is used by the aircraft with very important cuts for what concerns the costs of the airlines. [64]
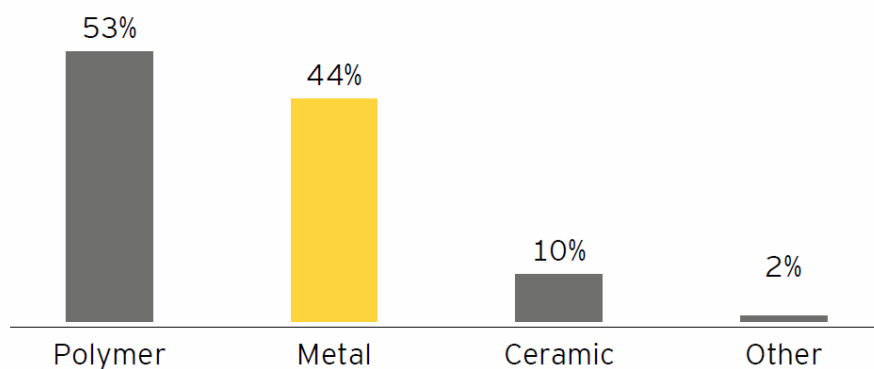

Figure 45: 3DP materials used (%). [65]

The graph above shows, instead, the percentage demand of 3D printing materials, as of 2016. Here, the leaders are the polymer materials followed shortly after by metals. This is due to the fact that companies have a lot of experience for what concerns the use of both categories, but much less use is registered for ceramics. Here, instead, the use is much lower. Only 10% of the companies make use of it and this is probably due, in the first place, to its durability and flexibility. Ceramics don´t allow a certain kind of item to be created like plastics or filaments do. Meanwhile, metals allow them make use of materials like titanium, which has a very strong resistance, steel and aluminium. [66]

Today's trends confirm the path that has been undertaken already. As of late 2017, it has been reported that PLA materials are currently at the top of the chart with a 32% of the total share. It is shortly after followed by ABS filaments, with 14% of the share. Standard Resins are reported to be the third most used material, with an 8% of the share. At the same time, what the trends are showing is that some materials have been used more than before due to the increase of use of a particular machine and therefore of a specific technology. In this case, for example, PA 12 material has been increasing in its use thanks to the new machines

for SLS technology that the market offers currently, alongside with HP's Multi Jet Fusion Technology. [67]

The following are instead some of the indicators regarding the current offer of 3D printing materials, including the amount of **bio-based** materials used, as well as the **colours** and the **diameters** available, alongside with many other characteristics.
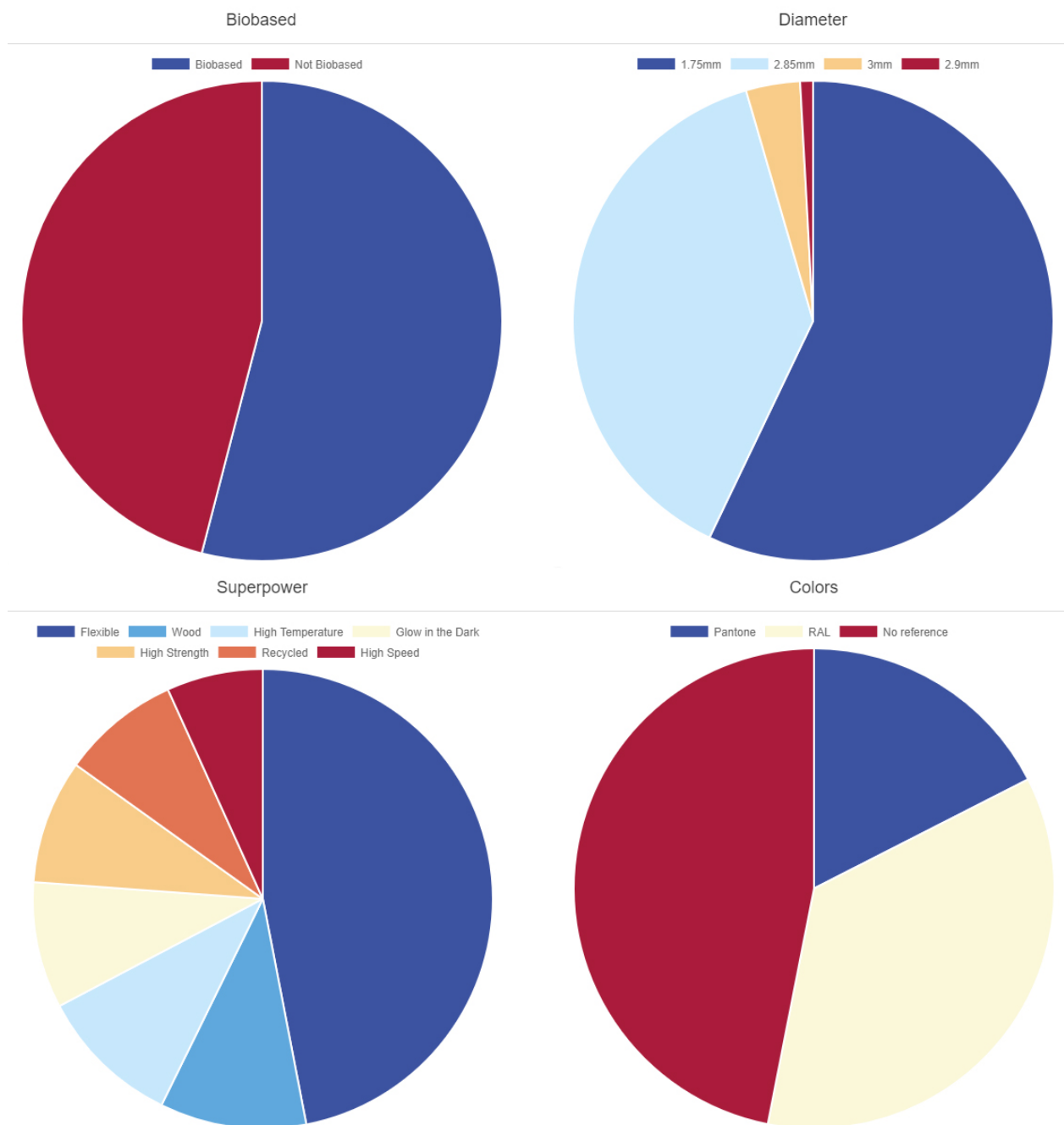

Figure 46: 3D Printer Filament Trends: November 2017. [68]

5.2. FDM, SLS and SLA 3D Printing Materials

According to the latest trends as of late November 2017, and based on the study conducted in chapter 2, the three most used 3D printing technologies are currently:

1. Fused Deposition Modeling (FDM);
2. Selective Laser Sintering (SLS);
3. Stereolithography (SLA).

For what concerns **Fused Deposition Modeling** (FDM), some of the most popular materials are:

- Thermoplastic Filament
- PLA: Easy to print, very accurate, low melting point, rigid. Good for most things, not for warm/hot regions.
- ABS: Tends to warp, strong and slightly flexible. Used for many mechanical parts.
- PETG: Easy to print & accurate, tends to string, good layer adhesion.
- TPU: Flexible polymer; ok to print, kind of like stiff rubber. Best used with direct-drive machines.
- Nylon: Strong and flexible; special filament for printers exist. Polycarbonate: Obnoxiously strong, high warp, high temp, bad fumes [Paul Chase, 3D Printing 101. 2016.]

Concerning ABS and PLA, there is no limit that one can do of them, for example they can be mixed together. They represent a much better solution to using pure plastic material; this is, in fact, relatively expensive, with a low strength and durability. Final products can usually be distorted easily. Therefore, plastics do not make FDM cost-effective, nor it allows the technology to be applied in functional and load-bearing applications, especially in a large-scale production.

ABS filaments, instead, can take a variety of forms and can be engineered to have many properties. Therefore, they are strong, but also very flexible; they can be sanded and machined, plus acetone can be used to gloss the parts through brushing or dipping them in it. A first comparison between ABS and PLA is that the former filaments are much easier to be recycled, another reason why most of engineers would rather prefer using this kind of material.
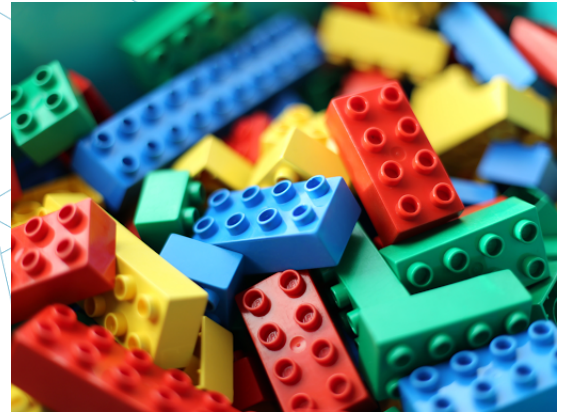

Figure 47: Lego bricks from ABS. [69]

For what concerns **Selective Laser Sintering** (SLS), the most popular materials are represented by the category of polyamides. A very clear example is nylon.

Objects, in this case, are created through an extruded filament or through sintered fine powder. The material is, in fact, rigid, strong, but it also flexes under high loads. Polyamides are also considered to be among the cheapest materials.

Post-processing prints created with this kind of material allows any kind of polish, dye or paint activities, and giving therefore extended freedom. The negative side of polyamides is that they absorb humidity quickly and need to be properly stored.

Figure 48: Polyamide. A strong and flexible material with a high level of detail. [70]

As for **Stereolithography**, plastics and resins are widely used. The market offers a variety of different resins which consist of different backbones and side groups—different combinations of long and short monomers, oligomers, photo initiators, and additives. This provides unique freedom to create various formulations with a wide range of optical, mechanical, and thermal properties, from clear to opaque and colored, flexible to rigid, and tough to heat resistant.

Resins can come in the form of liquid photopolymer materials, which are cured and hardened through ultraviolet (UV) energy. Resins are today one of the best materials in terms of possibility. The technologies that evolved around resins are the ones that offer the

highest quality, and this make it a great choice for professional applications, important product presentations, small objects, and so on.

Advantages in the use of resins include suitability for high resolution prints, smooth textures without the need of post-processing them, a wide variety of colours and properties, including transparency and translucency.

Disadvantages, instead, include the fact that resins can be irritating and toxic, leading to the need of the correct equipment in order to treat them. Cleaning is also required. Although the material offers a lot of freedom, the printing will require a support structure leading to an increase of material purchase and therefore it might become costly. This last matter is also influenced by the amount of post-processing steps that are undertaken.

6. LIMITATIONS OF PRINTED OBJECTS

In this section we aim to show the limitations of both additive manufacturing and 3D printing in general, as well as the particular limitations of each technology, focusing on the technology *Fused Deposition Modelling* (FDM).

6.1. Limitations of additive manufacturing and 3D printing

Despite the obvious advances that this technology can bring to the industry due to its indisputable advantages, there are limitations that make additive manufacturing technologies not widely implemented in many sectors.

The current limitations are due both to the AM processes themselves, which can still be improved, as well as to auxiliary processes (previous manipulations of material, post-processing, quality control ...) that in many cases condition their viability. The lack of knowledge on how to design the products and how to reorient the companies to successfully integrate these new technologies also influences. [71]

These limitations are undoubtedly surmountable and constitute challenges for research, technological development and innovation, which are currently being addressed by teams of researchers and by many companies around the world. Some of these limitations are the following:

Technologies in development: Many of the technologies are in continuous development. Some of them are still in early stages of development. In a few years, a machine may be outdated or outmoded.

Availability and cost of the raw material: Getting the supply of materials at some points can be complicated. Also, some materials are quite expensive, if you are looking for very specific properties. The most basic and cheap material is usually plastic.

Initial investment: Desktop FDM printers are usually cheap, but for other technologies, especially if larger machine sizes are sought, the initial investment can be very high.

Size of the pieces: The size of the pieces to produce depends on the printing bed. While it is true that many large pieces can be chopped and then joined.

Non-profitable mass production: Additive manufacturing and 3D printing are designed for very short series of pieces (around 1 - 10 units). For longer series, production with these technologies is not profitable.

Finishes and precision: In general, getting very good finish and very good precision will be expensive. That is to say, it will be necessary to resort to more expensive technologies and machines. In addition, the parts may require post-processes, other machines and special sub-processes.

Obtaining digital files: If specific and highly customized pieces are desired, it is necessary to possess advanced CAD design knowledge. Most of the time, the repositories of CAD files are not enough. The same goes for the 3D scanner; they are expensive and using it requires certain knowledge.

6.2. Limitations depending on the type of technology

Now certain limitations will be specified, grouped according to each type of technology:

• **Limitations of Fused deposition modeling (FDM).**

Poor surface finish and low print speed compared to other 3D printing technologies. The typical print size for desktop FDM printers is 20x20x20 cm. Requires support structures to print parts with lower angles of 45 degrees. The walls must have a minimum of 0.8 mm thickness. The engraved or embossed details can not be more accurate than 0.6 mm wide and 2 mm high. Horizontal "bridges" of more than 10 mm can not be made without the need

for supports. In general, holes smaller than 2 mm in diameter can not be printed. To ensure that a part of a piece does not fail when it is printed, the size of this part must be greater than 2 mm. When it comes to printing connection pins, the recommended minimum diameter is 3 mm. The tolerance (dimensional accuracy) expected is ± 0.5% (± 0.5 mm approximately).
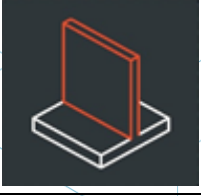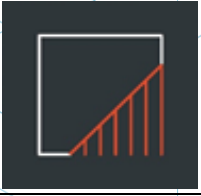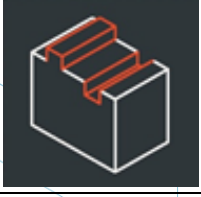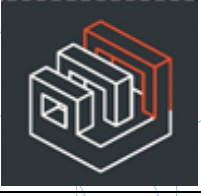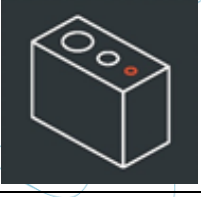
• **Limitations of selective laser sintering (SLS).**

The average build volume is around 30x30x30 cm. The finish is slightly granulated to the touch and matt, so if a glossy and smooth finish is desired, post-processing is recommended. The walls must have a minimum of 0.7 mm thickness. Embossed or engraved details should be at least 1mm (width and height). In general, holes with a smaller diameter than 1.5 mm can not be printed. To remove the support material (un-sintered dust), there must be leak holes in the parts. These must have a minimum diameter of 5 mm. The minimum size of a part of a piece so that the impression does not fail is about 0.8 mm, same for the connection pins. The expected tolerance is approximately ± 0.3% (± 0.3 mm).

• **Limitations of stereolithography (SLA).**

SLA printers generally have a much smaller build volume than most FDM printers, with the exception of industrial machines, which can have a print size of 14.5x14.5x14.5 cm. When the piece is larger, it is better to print it in smaller sections and then assemble them. The cost of the resins is very high (from € 150 / liter). Most parts printed by SLA need a post-curing process in a UV oven. The minimum recommended width of the walls is 0.5 mm (if the wall in question is connected to another element) or 1 mm (if it is not). Support structures are always required for cantilevered parts (this also makes printing more expensive). The engraved or embossed details should be 0.4 mm (width and height). The smallest holes that can be printed will be approximately 0.5 mm in diameter. To allow the material to come out in hollow pieces, escape holes should be left, which should be approximately 4 mm in diameter. So that an impression does not fail, the minimum size of the parts of the piece will be 0.2 mm. For connection pins, a diameter of 0.5 mm is recommended. The expected tolerance or dimensional accuracy is around ± 0.5% (± 0.15 mm).

COMPARATIVE TABLE

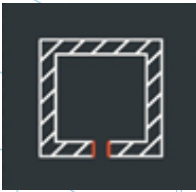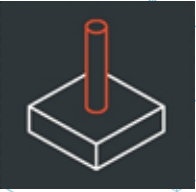| LIMITATION			TECHNOLOGIE		
NAME	DESCRIPTION	SKETCH	FDM	SLA	SLS
Supported Walls	Walls that are connected to the rest of the print on at least two sides.		0.8 mm	0.5 mm	0.7 mm
Unsupported Walls	Unsupported walls are connected to the rest of the print on less than two sides.		0.8 mm	1 mm	
Support & Overhangs	The maximum angle a wall can be printed at without requiring support.		45°	Support always required	
Embossed & Engraved Details	Features on the model that are raised or recessed below the model surface.		0.6 mm wide &2 mm high	0.4 mm wide & high	1 mm wide & high
Horizontal Bridges	The span a technology can print without the need for support.		10 mm		
Holes	The minimum diameter a technology can successfully print a hole.		2 mm	5 mm	1.5 mm
Connecting/Moving Parts	The recommended clearance between two moving or connecting parts.		0.5 mm	0.5 mm	0.3 mm for moving parts & 0.1 mm for connections

Escape Holes	The minimum diameter of escape holes to allow for the removal of build material.			4 mm	5 mm
Minimum Features	The recommended minimum size of a feature to ensure it will not fail to print.		2 mm	0.2 mm	0.8 mm
Pin Diameter	The minimum diameter a pin can be printed at.		3 mm	0.5 mm	0.8 mm
Tolerance	The expected tolerance (diameter accuracy) of a specific technology.		± 0.5% (lower limit ± 0.5 mm)	± 0.5% (lower limit ± 0.15 mm)	± 0.3% (lower limit ± 0.3 mm)

Table 4: Comparative table. [72]

6.3.   Introduction to the limitations of FDM technology

Going a little deeper into the FDM technology here we show some of its limitations in detail and propose some methods that can be implemented in the design stage to minimize the impact of these limitations when printing an object.

BRIDGING

Bridging in FDM occurs when the printer is required to print between two supports or anchor points. Because there is no support offered for the initial layer being printed (there is nothing to build upon) and it is required to "bridge" a gap, the material will tend to sag. Bridges occur most often in horizontal axis holes found in the walls of objects or in the top layer (or roof) of hollow parts.

One solution to reduce the impact of bridging is to reduce the distance of the bridge, but this will depend upon the design constraints of the part. Another solution to avoid sagging is to include support. Support offers a temporary build platform for the bridging layer to be built upon. The support material is then removed once the print has been completed. This can leave marks or damage upon the surface where the support was connected to the final part.

Figure 49: FDM printed puzzle piece with support removed showing surface roughness. [73]

Key design consideration: Due to the nature of FDM, sagging or marks from support material will always be present to some extent unless the bridge is less than 5mm.

An advanced solution is to split the design into separate parts or consider some form of post processing if a level and smooth surface is required.

VERTICAL AXIS HOLES

FDM will often print vertical axis holes undersized. The general process for printing a hole diameter and the reason the reduction in diameter occurs is:

1. As the nozzle prints the perimeter of a vertical axis hole, it compresses the newly printed layer down onto the existing build layers to help improve adhesion.
2. The compressing force from the nozzle deforms the extruded round layer shape from a circle into a wider and flatter shape (see image below).
3. This increases the area of contact with the previously printed layer (improving adhesion), but also increases the width of the extruded segment.
4. The result of this is a decrease in the diameter of the hole that is being printed.

This can be of particular issue when printing small diameter holes where the effect is greater due to the ratio of hole diameter to nozzle diameter.
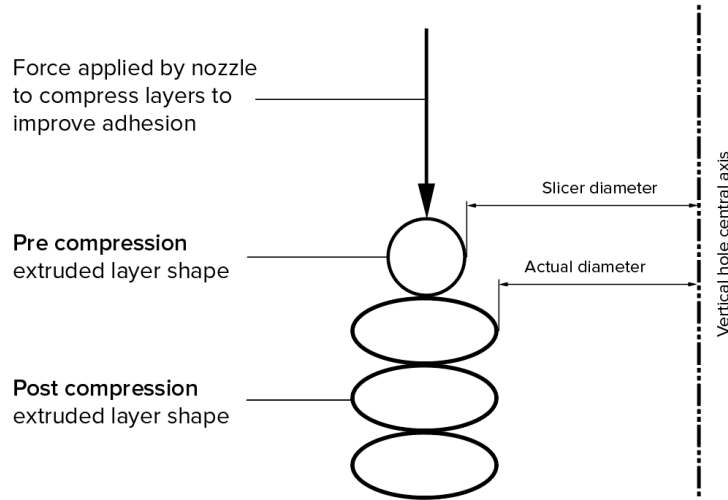

Figure 50: The variation in slicer program vs. actual diameter of vertical holes is due to compression of the extruded profile. [74]

The amount of undersize will depend on the printer, the slicing software, the size of the hole, and the material. Often, the reduction in diameter of vertical axis holes is accounted for in the slicing program, but accuracy can vary, and several test prints may be needed to achieve the desired accuracy. If a high level of accuracy is required, drilling the hole after printing may be required.

Key design consideration: If the diameter of your vertical axis hole is critical, printing it undersized and then drilling the hole to the correct diameter is recommended.

OVERHANGS

Issues with overhang are one of the most common print quality problems relating to FDM. Overhangs occur when the printed layer of material is only partially supported by the layer below. Similar to bridging, the inadequate support provided by the surface below the build layer can result in poor layer adhesion, bulging or curling.

Figure 51: The effect of increasing overhang angle (in increments of 5 degrees) on print quality. Max. angle shown is 70 degrees. [75]

An overhang can usually be printed with no loss of quality up to 45 degrees, depending on the material. At 45 degrees, the newly printed layer is supported by 50% of the previous layer. This allows sufficient support and adhesion to build upon. Above 45 degrees, support is required to ensure that the newly printed layer does not bulge down and away from the nozzle.

Another issue that occurs when printing overhangs is curling. The newly printed layer becomes increasingly thinner at the edge of the overhang, resulting in differential cooling causing it to deform upward (see the image above).

Key design consideration: Limitations on overhangs can be eliminated by the use of support for wall angles above 45 degrees. For larger overhangs where support is needed, marks will be present on the final surface unless post processed.

CORNERS

Because the printing nozzle in FDM is circular, corners and edges will have a radius that is equal to the size of the nozzle. This means that these features will never be perfectly square. For sharp edges and corners, the first layers of a print are particularly important. As discussed above for vertical holes, as the nozzle prints each layer, it compresses the print material down to improve adhesion. For the initial print layer, this creates a flare often called an

"elephant's foot". This can impact the ability to assemble FDM parts as this flare protrudes outside the specified dimensions.
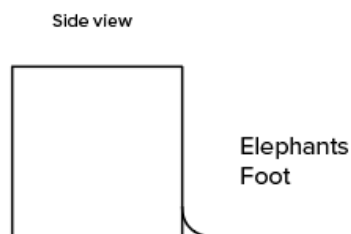

Figure 52: Side view of elephant's foot feature that can occur on the base layers of an FDM print. [76]

Another issue that is often present relating to the first print layer of an FDM print is warping. ABS is more vulnerable to warping due to its high printing temperature when compared to PLA. The base layer is the first layer to be printed and cools as the other hot layers are printed on top. This causes differential cooling and can result in the base layer curling up and away from the build plate as it shrinks and contracts.

The addition of a chamfer or radius along the edges of the part that are in contact with the build plate will reduce the impact of these problems. This will also assist in removal of the component from the build plate once the print has been completed.

Key design consideration: If assembly or overall dimensions are critical to the function of an FDM part, include a 45-degree chamfer or radius on all edges touching the build plate. For high precision form & fit testing, other technologies such as SLA or Polyjet are recommended.

VERTICAL PINS

Vertical pins are often printed in FDM when assembly of parts or alignment is required. Considering that these features are often functional, it is important to understand the size of vertical pins that FDM can print accurately.

Large pins (greater than 5mm diameter) are printed with a perimeter and infill, affording a strong connection to the rest of the print. Smaller diameter pins (less than 5mm diameter) can be made up of only perimeter prints with no infill. This creates discontinuity between the rest of the print and the pin, resulting in a weak connection that is susceptible to

breaking. In a worst-case scenario, small pins may not print at all as there is not enough print material for the newly printed layers to adhere to.

Often correct printer calibration (optimal layer height, print speed, nozzle temperature etc.) can reduce the likelihood of small pins failing. The addition of a radius at the base of the pin will eliminate that point as a stress concentration and add strength. For critical pins smaller than 5mm diameter, an off the shelf pin inserted into a printed hole may be the optimal solution.

Key design consideration: If your design contains pins smaller than 5 mm diameter, add a small fillet at the base of the pin. If function is critical, consider including a hole in your design in the location of the pin, drill the hole to the correct size and insert an off the shelf pin.

Figure 53: Print of vertical pins with decreasing diameter (from 25 to 5mm) illustrating the upper diameter of the print becoming too small to print accurately. [77]

ADVANCED DESIGN

Several key aspects to consider when printing with FDM are how to reduce the amount of support required, part orientation and the direction the part is built on the build platform.

Splitting up your model

Often, splitting up a model can reduce its complexity, saving on cost and time. Overhangs that require a large amount of support may be removed by simply splitting a complex shape into sections that are printed individually. If desired, the sections can be glued together once the print has been completed.
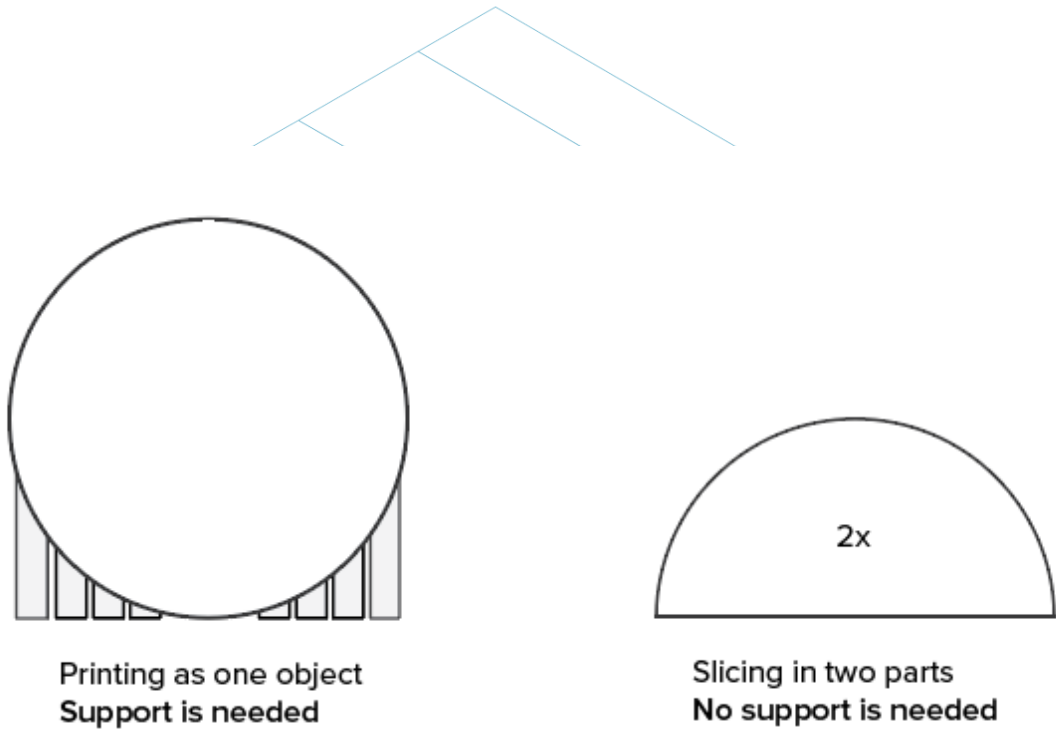
Figure 54: Splitting a model in order to eliminate the need for support. [78]

HOLE ORIENTATION

Support for holes is best avoided by changing the print orientation. Removal of support in horizontal axis holes can often be difficult, but by rotating the build direction 90 degrees, the need for support is eliminated. For components with multiple holes in different directions, prioritize blind holes, then holes with the smallest to largest diameter, then criticality of hole size.
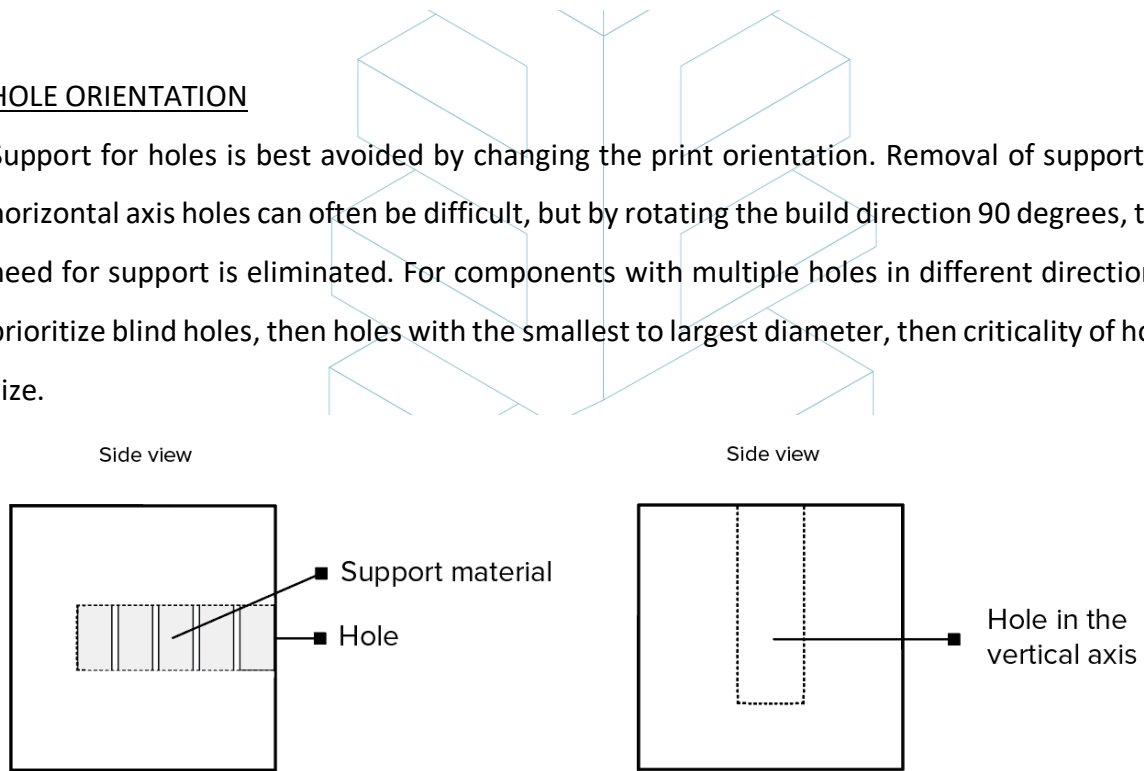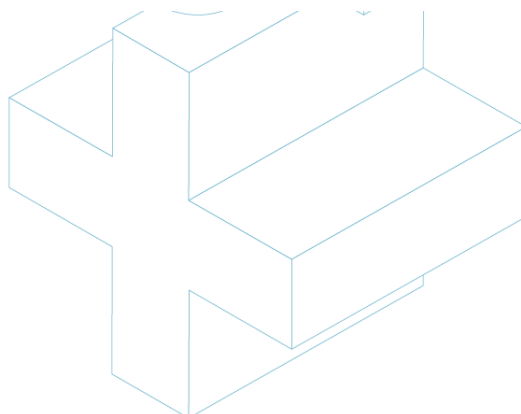

Figure 55: Re-orientation of horizontal axis holes can eliminate the need for support. [79]

BUILD DIRECTION

Due to the anisotropic nature of FDM printing, understanding the application of a component and how it is built are critical to the success of a design. FDM components are inherently weaker in one direction due to layer orientation.
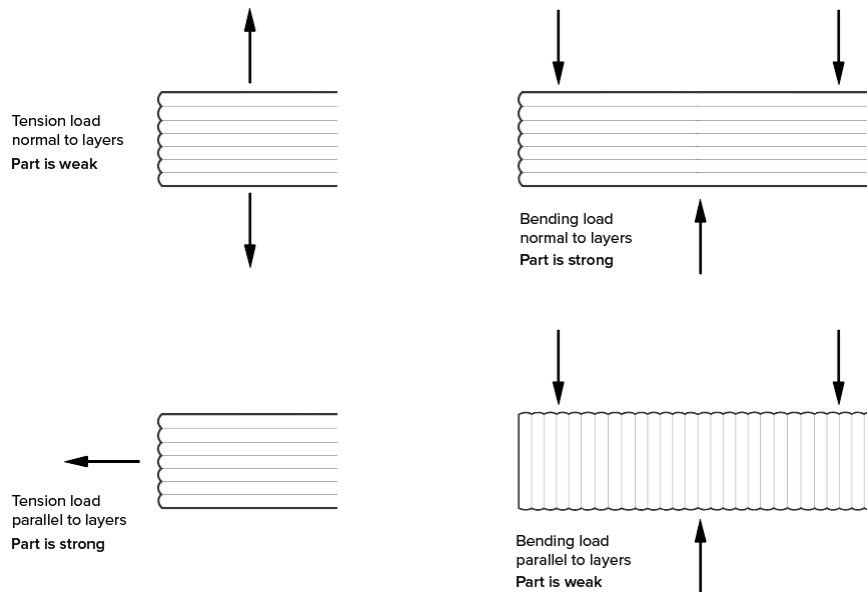

Figure 56: Build direction. [80]

The lack of continuous material paths and the stress concentration created by each layer joint contribute to this weakness. Since the layers are printed as a round-ended rectangle, the joints between each layer are actually small valleys. This creates a stress concentration where a crack will want to form.
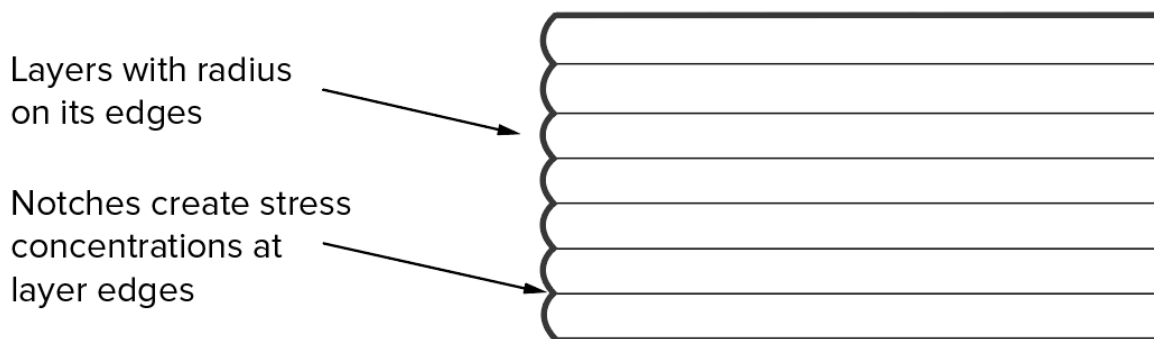

Figure 57: Layers with radius. [81]

RULES OF THUMB

- If a bridge exceeds 5mm, sagging or marks from support material can occur. Splitting the design or post processing can eliminate this issue.
- For critical vertical hole diameters, drilling after printing is recommended if high accuracy is desirable.
- The addition of support will allow FDM printers to print wall angles greater than 45 degrees.
- Include a 45-degree chamfer or radius on all edges of an FDM part touching the build plate.
- For applications with small vertical pins, add a small fillet at the base or consider inserting an off the shelf pin into a printed hole instead.
- Splitting a model, re-orientating holes and specifying build direction are all factors that can lower cost, speed up the printing process, and improve the strength and print quality of a design. [82]

7. EXAMPLE

In this chapter it is intended to show, by a real example, all the steps to follow for, starting from a model or digital file, getting a real printed piece.

The example shown hereunder is about a heritage reproduction project. It arises from the need to make certain pieces of the Museum of Natural History of Valencia accessible to blind people. Specifically, the project is about the lower part of a tiger jaw. The project begins with the 3D scanning of the original jaw, getting a digital model then. This example will begin from this point.
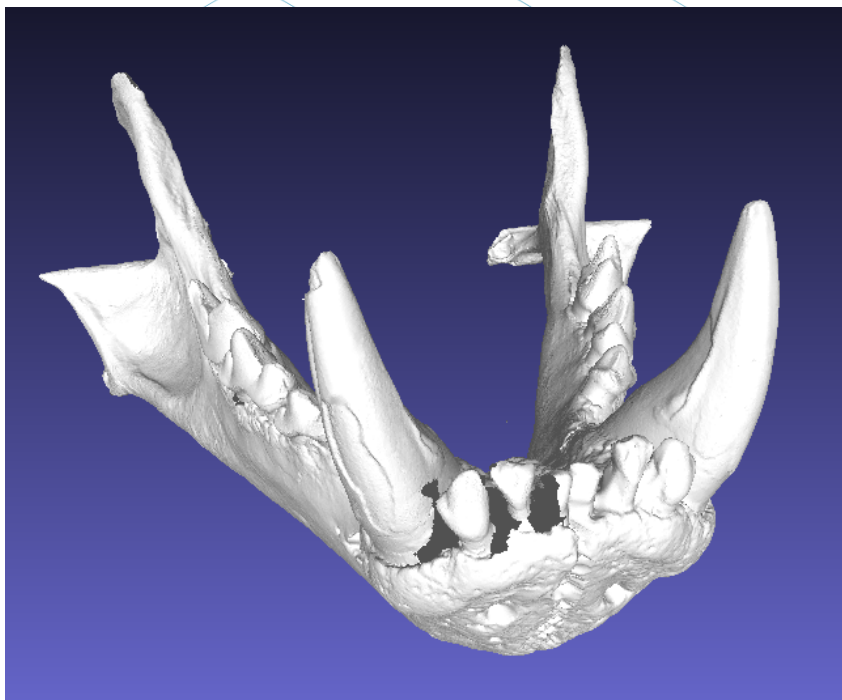
Figure 58: Digital model, result of the 3D scanning. [83]

Whatever it is the file format, it has to be exported to STL, as it has been shown in other chapters of this guide.

As it can be seen almost simple sight in the previous Figure, the model has several errors that need to be repaired, like the teeth part, for example.

For that, several software can be used. The process of analysis and repairing will be explained here with free-access software. For example: Meshmixer, from Autodesk.

Before seeing the process of analysis and reparation of a STL model, it is noted that, this model under consideration was processed by a certain software (Meshlab, specifically) in order to reduce the file size. The model mesh had a very high and excessive number of triangles, so this number was lowed and hence, the file size. This is a step that just a few times is necessary to implement, so it is just mentioned. For further information about how to reduce the number of triangles of a mesh, and hence the size file, the following link is provided:

https://www.shapeways.com/tutorials/polygon_reduction_with_meshlab

So, as it was said before, the STL model will be opened with Meshmixer, by clicking on Import. The first thing that can be seen is that the model orientation could not be the adequate one. There is no need to be worry about that. Meshmixer allows you to change it

by clicking on Edit > Transform. In addition, we will not print by Meshmixer, so orientation has been adjusted approximately, as can be seen here:
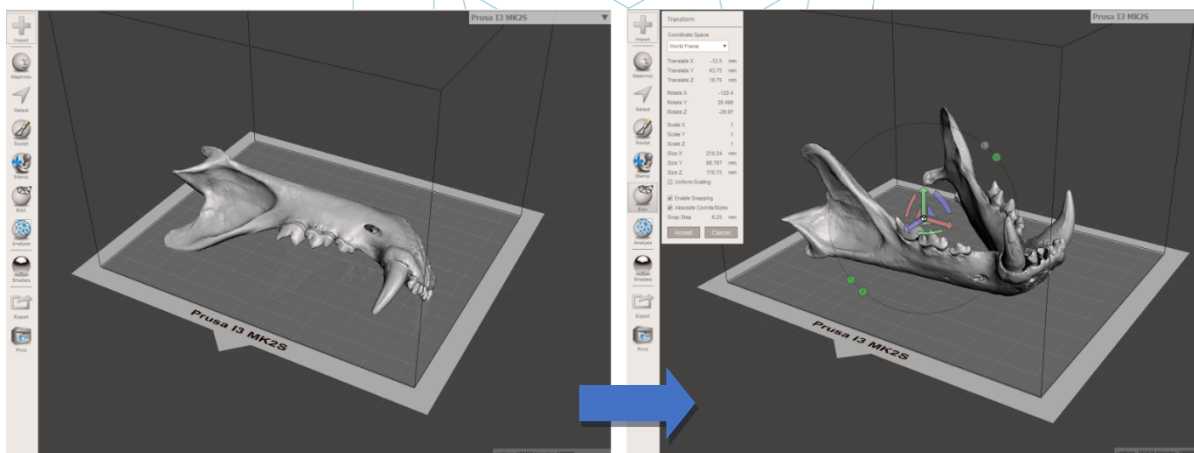

Figure 59: Positioning the model in Meshmixer. [84]

Thereupon, a general analysis of the defects of the piece is made. For that, Inspector option is used, in Analysis section. Meshmixer detects and shows all the errors in the model or file. The different mistakes are shown in different colours, and they are shown by balls and arrows. Holes in the mesh are displayed in blue, non-manifold points or edges are displayed in red, and disconnected components or elements with a very little area are displayed in magenta. There are two options now, either repairing the errors one by one by clicking on the "ball", or either clicking on Auto Repair All and repairing everything at once.
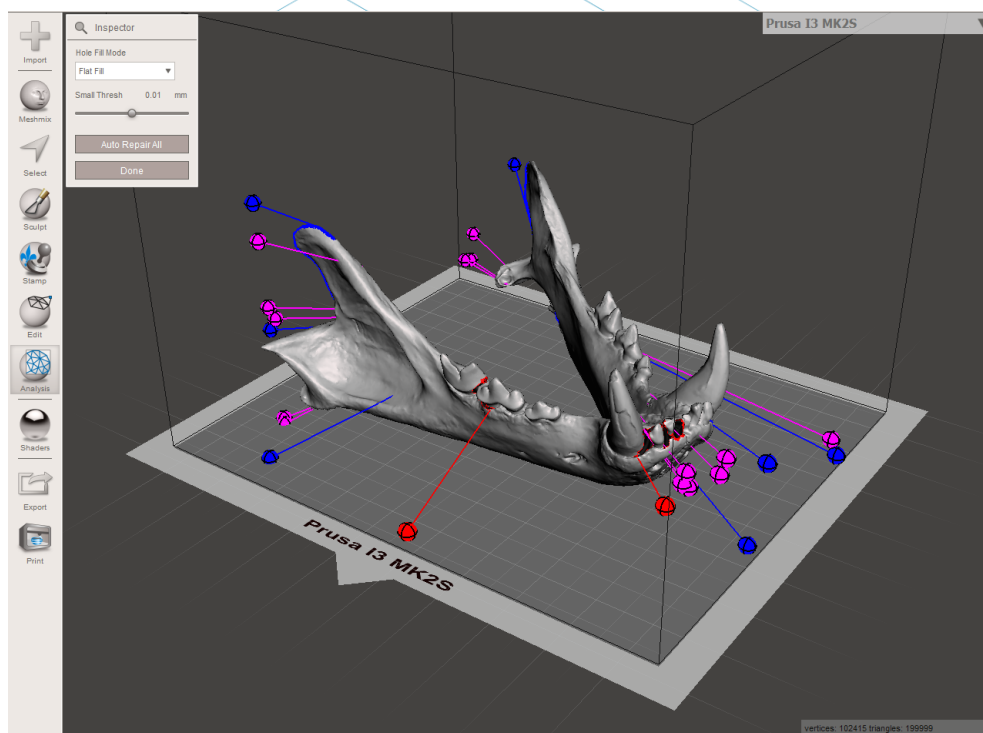
Figure 60: General analysis in Meshmixer. [85]

In this case, the software repairs all the found defects automatically. If not, it is possible to use other software and implement other analysis and reparation process.

With Meshmixer it is also possible to perform other kind of analysis such as: thickness, stability, orientation and positioning on the printing base, etc.

So, at this point, the STL model is completely repaired and it is ready for the next step. As it can be seen, the program has closed every hole in the mesh, and it has repaired all the defects. It is noted that the software does this reparation automatically, trying to get close, more or less, to the geometry around the defect. If holes or gaps in the mesh are intended to be repaired with a certain geometry it is needed to use a digital sculpture software. It is true that Meshmixer has a digital sculpture option, but it will not be explained in this example, because it is not an objective of this guide.
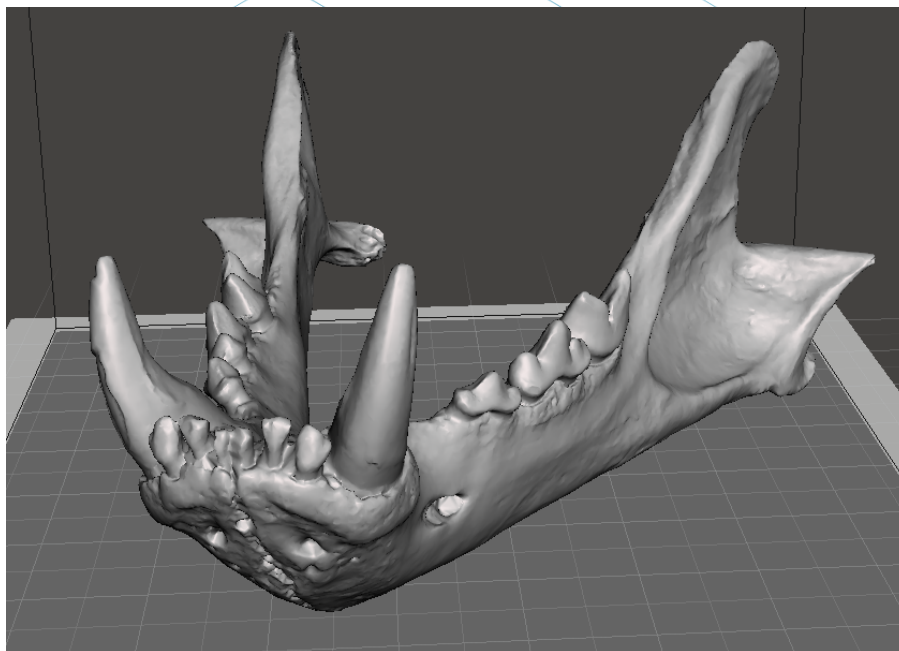
Figure 61: Jaw with repaired holes and defects- [86]

So now, in Meshmixer, the repaired and fixed model must be exported or saved again in STL format.

The next step is to load the file in a software that allows to do, at least, the following things:

- Choosing an orientation in the printing bed. Rotating and moving the model.
- Scaling the model.
- Choosing the quality or layer height.
- Choosing the infill. Density among other issue.
- Generating the needed support structures.

In addition, with this software the G-Code or machine code will be also generated. There is different free software for doing this. For example, Ultimaker Cura or Slic3r. In this example Cura is used.

So, in Cura the repaired model is imported. To configure the software in order to show the printing bed of the machine now is convenient. Cura has a very big data base of commercial 3D printers. If the used machine is not in that data base, it is also possible to create it from scratch.
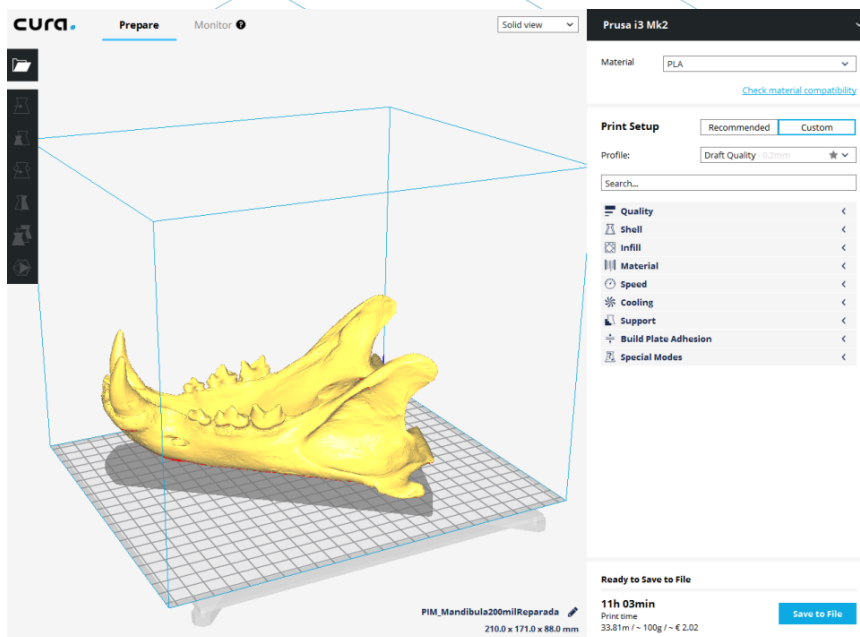
Figure 62: Model loaded in Cura. [87]

This model is going to be scaled to 60%. 0.2 mm. is the chosen layer quality. A grid infill of 15% density is also chosen. Automatically support structures will be generated everywhere, and in order to get a good adhesion for the first layer, "first layer speed" will be set to 20 mm/s. working material is also chosen: PLA. Finally, the G-Code is saved in the device to insert in the printer by clicking on "Save to File".
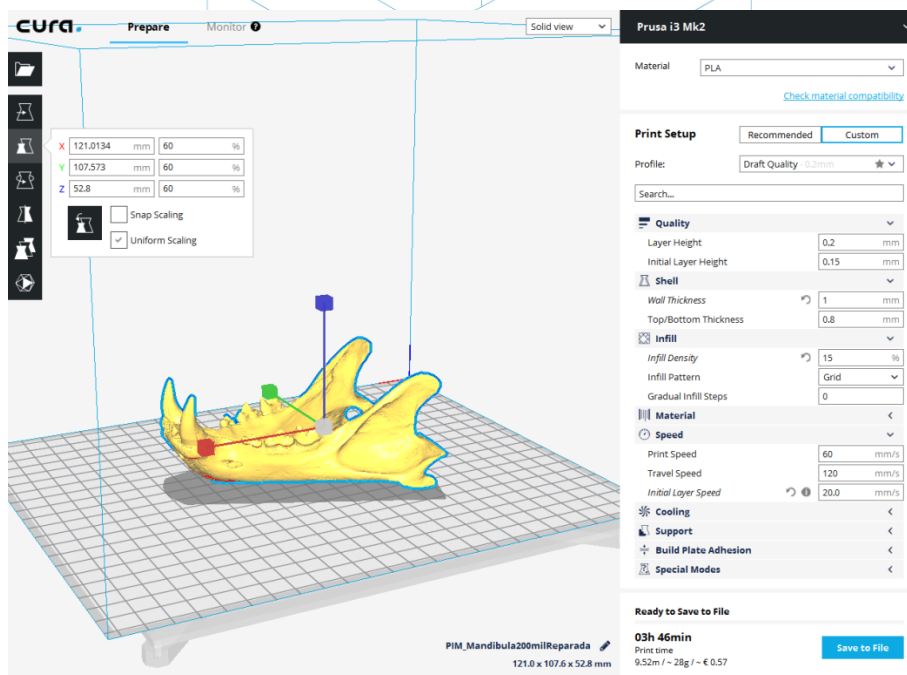

Figure 63: Configuration and parameters of the model in Cura. [88]

At this point, it is possible to go to the next step of the process: 3D printing itself. It is needed to go to the machine and, by the appropriated means, to launch the print job. The machine will be left working, and it is recommended to see it from time to time in order to see if everything goes correctly. It is important to say that it is highly recommended to check if the first printed layer is correctly stick to the bed or platform. If not, the following guide is provided: https://www.simplify3d.com/support/print-quality-troubleshooting/#print-not-sticking-to-the-bed

In the next picture the printing sequence is shown, for the given example:
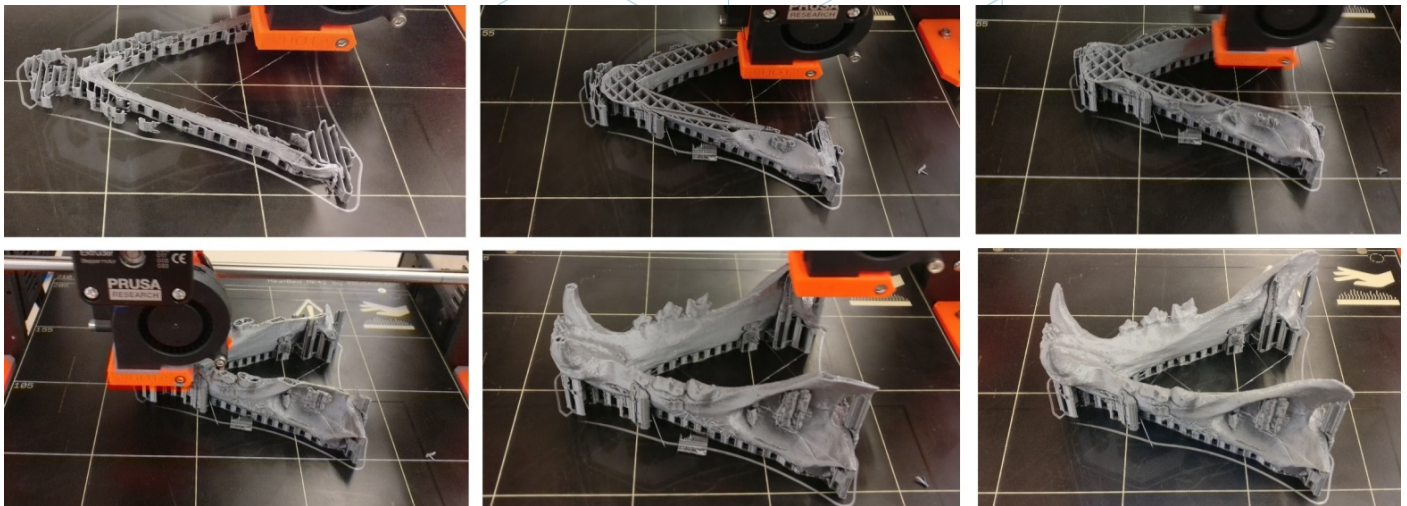

Figure 64: Jaw printing sequence. [89]

The next step is, obviously, to extract the printed piece. A spatula with rounded edges is used, in order not to scratch and not to damage the printing platform.

Figure 65: Piece extraction process. [90]

Once the piece has been extracted of the platform, it is time for the post-processing step, consisting on manually and mechanically removing the support structures, and sanding those parts of the piece that have marks (from support structures, for example). The printing bed should be correctly cleaned for the next print job.
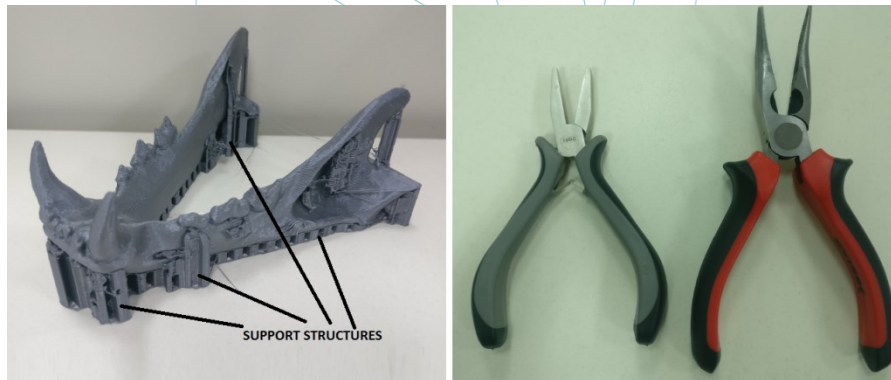

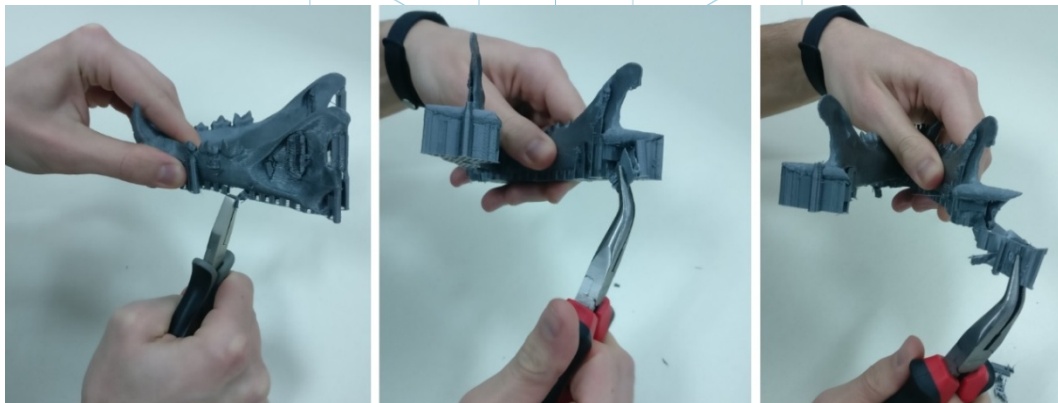*Figure 66: Support structures and pliers. [91]*

Figure 67: Removing support structures. [92]

As mentioned, to sand the marks of the support structures, among others, is very convenient.

At the end, the final result of the piece is the following one:

Figure 68: Final result. [93]

If the result at any point of the printing process, or at the end of the printing process is not good, or some errors and shrinkages are seen, the following guide can be very helpful, in order to configure some printing parameters or parameters of the printer:

https://www.simplify3d.com/support/print-quality-troubleshooting/

SOURCES

[1] What is Additive Manufacturing? Source: _https://www.3dhubs.com/what-is-3d-printing_

[2] Scheme 1: Processes and manufacturing techniques. Source: _KIT_

[3] Figure 1: FDM Printer. Source: _https://proto3000.com/assets/uploads/Images/ProductImages/MB05_REP_01B_Hero.png_

[4] Figure 2: SLA Printer. Source: _http://www.rs-online.com/designspark/rel-assets/ds-assets/uploads/images/5630ebfd9a5c4cd48a521dba0ab5c436Formlabs%20Double%20Image.png_

[5] Figure 3: SLS Printer. Source: _https://www.3dsystems.com/sites/default/files/styles/thumbnail_twothirds_size/public/2017-02/sPro_140_Angle_940px_tn.png?itok=KOJ3Xo_w_

[6] How does 3D printing work? Source: _https://www.createeducation.com/about-us/introduction/_

[7] Figure 4: 3D printing process. Source: _KIT_

[8] Rapid prototyping technology for the development of new products. 2016. Source: _http://ijiset.com/vol3/v3s1/IJISET_V3_I1_39.pdf_

[9] Figure 5: Prototypes of a ski goggles' frame printed with FDM, SLA and SLS technology (from left to right). Source: _https://formlabs.com/de/blog/fdm-vs-sla-vs-sls-how-to-choose-the-right-3d-printing-technology/_

[10] Figure 6: 3D Printing Technologies Comparison. Source: _STP_

[11] Universe Berkeley: Mechanical Engineering Student Access Machine Shop: Stratus Dimensions Fused Deposition Modelling.

[12] Figure 7: FDM Technology. Source: _https://i.materialise.com/blog/3d-printing-technologies-and-materials/_

[13] F. Kunz, A. A. Jorg, L. Chaabane: Innovation Infrastructure& Services. Switzerland Innovation Park Biel. 2017.

[14] FDM Prints. 2017. Source: _https://www.sculpteo.com/en/glossary/fdm-fused-deposition-modeling-definition/_

[15] Surface modification of fused deposition modeling ABS to enable rapid prototyping of biomedical microdevices. 2013. Source: _http://www.sciencedirect.com/science/article/pii/S092401361300006X_

[16] Scaffold Design and in Vitro Study of Osteochondral Coculture in a Three-Dimensional Porous Polycaprolactone Scaffold Fabricated by Fused Deposition Modeling. 2004. Source: _http://online.liebertpub.com/doi/abs/10.1089/10763270360697012_

[17]FDM Technology 3D print durable parts with real thermoplastic. 2017. Source: http://www.stratasys.com/3d-printers/technologies/fdm-technology

[18]Figure 8: FDM Advantages. 2016. Source: http://www.materialise.com/en/manufacturing/3d-printing-technology/fused-deposition-modeling

[19]Figure 9: SLS Technology. 2017. Source: https://en.wikipedia.org/wiki/Selective_laser_sintering#Technology

[20]Selective laser sintering. 2017. Source: https://en.wikipedia.org/wiki/Selective_laser_sintering#Technology

[21]How Does Powder-Based 3D Printing Work? 2016. Source: https://imaterialise.helpjuice.com/design-printing/powder-based-3d-printing

[22]3D Printing Technology/Types of 3D Printers: SLS. 2016. Source: https://all3dp.com/types-of-3d-printer-technology-explained/#SLM

[23]Figure 10: Shoe silhouette created with SLS. Source: http://ibdesigntech.com/3-5-rapid-prototyping-selective-laser-sintering-sls-5/

[24]Figure 11: Resin 3D Printer parts overview. Source: https://www.tth.com/wp-content/uploads/SLA-Process.jpg

[25]Figure 12: Resins, properties. Source: https://all3dp.com/1/best-resin-dlp-sla-3d-printer-kit-stereolithography/

[26]Figure 13: Photocentric 3D prints.

[27]Scheme 2: Production process in 3D printing. Source: CETEM

[28]Figure 14: Piece modelled with Rhinoceros. Source: https://www.sculpteo.com/es/tutoriales/preparar-su-modelo-para-la-impresion-3d-con-rhinoceros/

[29]Figure 15: Reverse engineering process. Source: CETEM

[30]Figure 16: Examples of pieces from Thingiverse repository. Source: https://www.thingiverse.com/

[31]Figure 17: How STL file works. Source: https://createc3d.com/wp-content/uploads/2014/12/emendo.png

[32]Figure 18: Exporting to STL in different software. Source: https://www.sculpteo.com/en/tools/transferring-3d-file/

[33]Figure 19: Thickness analysis (1 millimetre) implemented with Meshmixer. Source: CETEM

[34] Figure 20: Angle analysis. Source: *CETEM*

[35] Figure 21: Support structures. Source: https://www.geeky-gadgets.com/wp-content/uploads/2015/01/Form-1-3D-Printer-Update1.jpg

[36] Figure 22: Different types of support structures. Source: *http://www.reppersdelight.spacymen.com/public/RD_images/MeshMixer_support3.jpg https://i.ytimg.com/vi/GThbJZZvRFY/maxresdefault.jpg https://www.simufact.com/files/Medien/_2Produkte/2.3_Simufact_Additive/Support-settings_more_support.png*

[37] Figure 23: Differences between a piece with optimized support structures (left) and non-optimized (right). Differences between printing times can also be seen. Source: *https://www.3ders.org/images/skin-frame-structure-3d-printing-material-6.png*

[38] Figure 24: Different infill percentages. Source: http://my3dmatter.com/wp-content/uploads/2015/03/infillpercentimage.png *https://3dplatform.com/wp-content/uploads/2015/08/3DP-Infill-Percentage-1024x745.jpg*

[39] Figure 25: Different infill patterns. Source: https://i.ytimg.com/vi/BMWTK2ZgJCM/maxresdefault.jpg https://assets.pinshape.com/uploads/image/file/98145/container_display-tray-for-infill-pattern-and-infill-density-3d-printing-98145.jpg

[40] Figure 26: Quality differences in the same piece, according orientation. Source: *https://s3-eu-west-1.amazonaws.com/3dhubs-knowledgebase/print-orientation/photo2.jpg*

[41] Figure 27: Tension load. Source: *https://s3-eu-west-1.amazonaws.com/3dhubs-knowledgebase/print-orientation/visual2.png*

[42] Figure 28: Different layers thickness. Source: *https://www.3dhubs.com/knowledge-base/impact-layer-height-3d-print*

[43] Figure 29: Two different layer heights. Source: *http://wiki.ikaslab.org/images/thumb/8/8c/Altura_de_capa.jpg/700px-Altura_de_capa.jpg*

[44] Figure 30: Detail of the layers of a piece. Source: *https://i.ytimg.com/vi/-KS_6P8tHZ4/maxresdefault.jpg*

[45] Figure 31: Layers and path. Source: *http://domoticx.com/wp-content/uploads/cura-screen.jpg*

[46] Figure 32: Extraction. Source: *https://i.ytimg.com/vi/1T5BdRFlCd8/maxresdefault.jpg https://d3v5bfco3dani2.cloudfront.net/photo/image/1300x0/58ee1eb0f407c/SDB2017-04-11_0001-61.JPG*

[47] Figure 33: SLA extraction process. Source: *https://www.3dnatives.com/es/wp-content/uploads/sites/4/SLA_Technology.jpg*

[48] Figure 34: SLS extraction process. Source: *https://3dprint.com/wp-content/uploads/2016/04/Untitled48.png* *http://www.oceanz.eu/files/original/640/sls-professional-3d-printing-oceanz.jpg*

[49] Figure 35: Finishing process. Source: *https://s3-eu-west-1.amazonaws.com/3dhubs-knowledgebase/fdm_post_processing/photo2.png*

[50] Figure 36: Mechanical process. Source: *https://formlabs.com/media/upload/_thumbs/Removing_Supports_copy.jpg.895x0_q80_crop-smart.jpg*

[51] Figure 37: Bath for removing the support structures. Source: *https://3dprint.com/wp-content/uploads/2015/07/rinse3.png*

[52] Figure 38: Different finishing. Source: *https://hackadaycom.files.wordpress.com/2013/02/rap.jpg?w=580*

[53] Scheme 3: Production process for 3D printing. Source: *CETEM*

[54] Figure 39: Process to develop a 3D Model. Source: *KIT*

[55] Table 1: Software to Designing a 3D Model. Source: *KIT*

[56] Table 2: Software to Scanning a 3D Model. Source: *KIT*

[57] Table 3: Software to Testing, Orientating and Repair a 3D Model. Source: *KIT*

[58] Figure 40: Repairing a STL file. Source: *KIT*

[59] Figure 41: Netfabb Premium Interface. Source. Source: *KIT*

[60] Figure 42: Open a 3D design with Slic3r. Source: *KIT*

[61] Figure 43: G-Code with Slic3r. Source: *KIT*

[62] Figure 44: Diagram 3D Printing Workflow. Source: *KIT*

[63] Print Materials. 2017. Source: *https://www.3dhubs.com/materials*

[64] How will 3D printing make your company the strongest in the value chain?. 2006. Source: *http://www.ey.com/Publication/vwLUAssets/ey-global-3d-printing-report-2016-fullreport/$FILE/ey-global-3d-printing-report-2016-full-report.pdf*

[65] Figure 45: 3DP materials used (%). Source: *http://www.ey.com/Publication/vwLUAssets/ey-global-3d-printing-report-2016-fullreport/$FILE/ey-global-3d-printing-report-2016-full-report.pdf*

[66] Explanation of the graphic. Source: _http://www.ey.com/Publication/vwLUAssets/ey-global-3d-printing-report-2016-fullreport/$FILE/ey-global-3d-printing-report-2016-full-report.pdf_

[67] 3D Printing Trends Q4/2017. 2017. Source: _https://f.3dhubs.com/yZgXoWzB88BhMHwG9fo3mV.pdf_

[68] Figure 46: 3D Printer Filaments Trends 2017. Source: _https://www.filaments.directory/en/trends_

[69] Figure 47: Lego bricks from ABS. Source: _https://www.matterhackers.com/articles/how-to-succeed-when-printing-with-abs_

[70] Figure 48: Polyamide. A strong and flexible material with a high level of detail. Source: _http://3dprintedinstruments.wikidot.com/materials#toc24_

[71] Limitation in 3D printing. Source: _http://informecotec.es/media/N30_Fabric_Aditiva.pdf_

[72] Table 4: Comparative table. Source: _https://s3-eu-west-1.amazonaws.com/3dhubs-knowledgebase/key-design-considerations-for-3d-printing/3d-printing-design-rules.jpg_

[73] Figure 49: FDM printed puzzle piece with support removed showing surface roughness. Source: _https://s3-eu-west-1.amazonaws.com/3dhubs-knowledgebase/key-design-considerations-fdm-3d-printing/photo14.jpg_

[74] Figure 50: The variation in slicer program vs. actual diameter of vertical holes is due to compression of the extruded profile. Source: _https://s3-eu-west-1.amazonaws.com/3dhubs-knowledgebase/key-design-considerations-fdm-3d-printing/visual20.png_

[75] Figure 51: The effect of increasing overhang angle (in increments of 5 degrees) on print quality. Max. angle shown is 70 degrees. Source: _https://s3-eu-west-1.amazonaws.com/3dhubs-knowledgebase/key-design-considerations-fdm-3d-printing/photo13.jpg_

[76] Figure 52: Side view of elephant's foot feature that can occur on the base layers of an FDM print. Source: _https://s3-eu-west-1.amazonaws.com/3dhubs-knowledgebase/key-design-considerations-fdm-3d-printing/visual6.png_

[77] Figure 53: Print of vertical pins with decreasing diameter (from 25 to 5mm) illustrating the upper diameter of the print becoming too small to print accurately. Source: _https://s3-eu-west-1.amazonaws.com/3dhubs-knowledgebase/key-design-considerations-fdm-3d-printing/photo15.jpg_

[78] Figure 54: Splitting a model in order to eliminate the need for support. Source: _https://s3-eu-west-1.amazonaws.com/3dhubs-knowledgebase/key-design-considerations-fdm-3d-printing/visual10.png_

[79] Figure 55: Re-orientation of horizontal axis holes can eliminate the need for support. Source: _https://s3-eu-west-1.amazonaws.com/3dhubs-knowledgebase/key-design-considerations-fdm-3d-printing/visual18.png_

[80] Figure 56: Build direction. Source: _https://s3-eu-west-1.amazonaws.com/3dhubs-knowledgebase/key-design-considerations-fdm-3d-printing/visual13.png_

[81] Figure 57: Layers with radius. Source: _https://s3-eu-west-1.amazonaws.com/3dhubs-knowledgebase/key-design-considerations-fdm-3d-printing/visual9.png_

[82] How to design parts for FDM 3D Printing. Source: _https://www.3dhubs.com/knowledge-base/how-design-parts-fdm-3d-printing_

[83] Figure 58: Digital model, result of the 3D scanning. Source: _CETEM_